P

Ma, John

From: Ma, John /NRO
Sent: Friday, January 22, 2010 8:21 AM
To: (b)(6)
Subject: FW: ABAQUS Concrete Model
Attachments: ABAQUS Concrete Material Models.doc

Ex. 4

From: Park, Sunwoo
Sent: Thursday, January 21, 2010 6:07 PM
To: Ma, John; Tegeler, Bret
Cc: Patel, Pravin; Thomas, Brian
Subject: ABAQUS Concrete Model

John, Bret,

I've collected some concrete material models used in the current ABAQUS code (v.6.9) for possible help with your on-going Shield Bldg analysis review. I was using ABQUS for some time involving some of its concrete models in the past, and still keep some material on it. (I need to refresh my memory to recollect them, though).

Lately, I've found that there is a site available where you can visit and view the ABAQUS Manuals on line (for free). Anyway, the site link is

http://opportunity.neu.edu/opportunity-docs/abaqus/v6.9/index.html

I've visited this site and downloaded some relevant concrete models and put them in the attached Word file. You will immediately see that some sketches of the models are identical to those presented in the WH package.

Sunwoo Park
STRUCTURAL ENGINEER
NRO/DE/SEB1
ROOM: T-10A7
(301) 415-2690

J/S

Abaqus 6.9 Analysis User's Manual

Part V: Materials

- Chapter 17, "Materials: Introduction"
- Chapter 18, "Elastic Mechanical Properties"
- Chapter 19, "Inelastic Mechanical Properties"
- Chapter 20, "Progressive Damage and Failure"
- Chapter 21, "Hydrodynamic Properties
- Chapter 22, "Other Material Properties"

19. Inelastic Mechanical Properties

- "Overview," Section 19.1
- "Metal plasticity," Section 19.2
- "Other plasticity models," Section 19.3
- "Fabric materials," Section 19.4
- "Jointed materials," Section 19.5
- "Concrete," Section 19.6
- "Permanent set in rubberlike materials," Section 19.7

19.6 Concrete

- "Concrete smeared cracking," Section 19.6.1
- "Cracking model for concrete," Section 19.6.2
- "Concrete damaged plasticity," Section 19.6.3

19.6.1 Concrete smeared cracking

Products: Abaqus/Standard Abaqus/CAE

References

- "Material library: overview," Section 17.1.1
- "Inelastic behavior," Section 19.1.1
- *CONCRETE
- *TENSION STIFFENING
- *SHEAR RETENTION
- *FAILURE RATIOS
- "Defining concrete smeared cracking" in "Defining plasticity," Section 12.9.2 of the Abaqus/CAE User's Manual

Overview

The smeared crack concrete model in Abaqus/Standard:

- provides a general capability for modeling concrete in all types of structures, including beams, trusses, shells, and solids;
- can be used for plain concrete, even though it is intended primarily for the analysis of reinforced concrete structures;
- can be used with rebar to model concrete reinforcement;
- is designed for applications in which the concrete is subjected to essentially monotonic straining at low confining pressures;
- consists of an isotropically hardening yield surface that is active when the stress is dominantly compressive and an independent "crack detection surface" that determines if a point fails by cracking;
- uses oriented damaged elasticity concepts (smeared cracking) to describe the reversible part of the material's response after cracking failure;
- requires that the linear elastic material model (see "Linear elastic behavior," Section 18.2.1) be used to define elastic properties; and
- cannot be used with local orientations (see "Orientations," Section 2.2.5).

See "Inelastic behavior," Section 19.1.1, for a discussion of the concrete models available in Abaqus.

Reinforcement

Reinforcement in concrete structures is typically provided by means of rebars, which are one-dimensional strain theory elements (rods) that can be defined singly or embedded in oriented surfaces. Rebars are typically used with metal plasticity models to describe the behavior of the rebar material and are superposed on a mesh of standard element types used to model the concrete.

With this modeling approach, the concrete behavior is considered independently of the rebar. Effects associated with the rebar/concrete interface, such as bond slip and dowel action, are modeled approximately by introducing some "tension stiffening" into the concrete modeling to simulate load transfer across cracks through the rebar. Details regarding tension stiffening are provided below.

Defining the rebar can be tedious in complex problems, but it is important that this be done accurately since it may cause an analysis to fail due to lack of reinforcement in key regions of a model. See "Defining reinforcement," Section 2.2.3, for more information regarding rebars.

Cracking

The model is intended as a model of concrete behavior for relatively monotonic loadings under fairly low confining pressures (less than four to five times the magnitude of the largest stress that can be carried by the concrete in uniaxial compression).

Crack detection

Cracking is assumed to be the most important aspect of the behavior, and representation of cracking and of postcracking behavior dominates the modeling. Cracking is assumed to occur when the stress reaches a failure surface that is called the "crack detection surface." This failure surface is a linear relationship between the equivalent pressure stress, p, and the Mises equivalent deviatoric stress, q, and is illustrated in Figure 19.6.1–5. When a crack has been detected, its orientation is stored for subsequent calculations. Subsequent cracking at the same point is restricted to being orthogonal to this direction since stress components associated with an open crack are not included in the definition of the failure surface used for detecting the additional cracks.

Cracks are irrecoverable: they remain for the rest of the calculation (but may open and close). No more than three cracks can occur at any point (two in a plane stress case, one in a uniaxial stress case). Following crack detection, the crack affects the calculations because a damaged elasticity model is used. Oriented, damaged elasticity is discussed in more detail in "An inelastic constitutive model for concrete," Section 4.5.1 of the Abaqus Theory Manual.

Smeared cracking

The concrete model is a smeared crack model in the sense that it does not track individual "macro" cracks. Constitutive calculations are performed independently at each integration point of the finite element model. The presence of cracks enters into these calculations by the way in which the cracks affect the stress and material stiffness associated with the integration point.

Tension stiffening

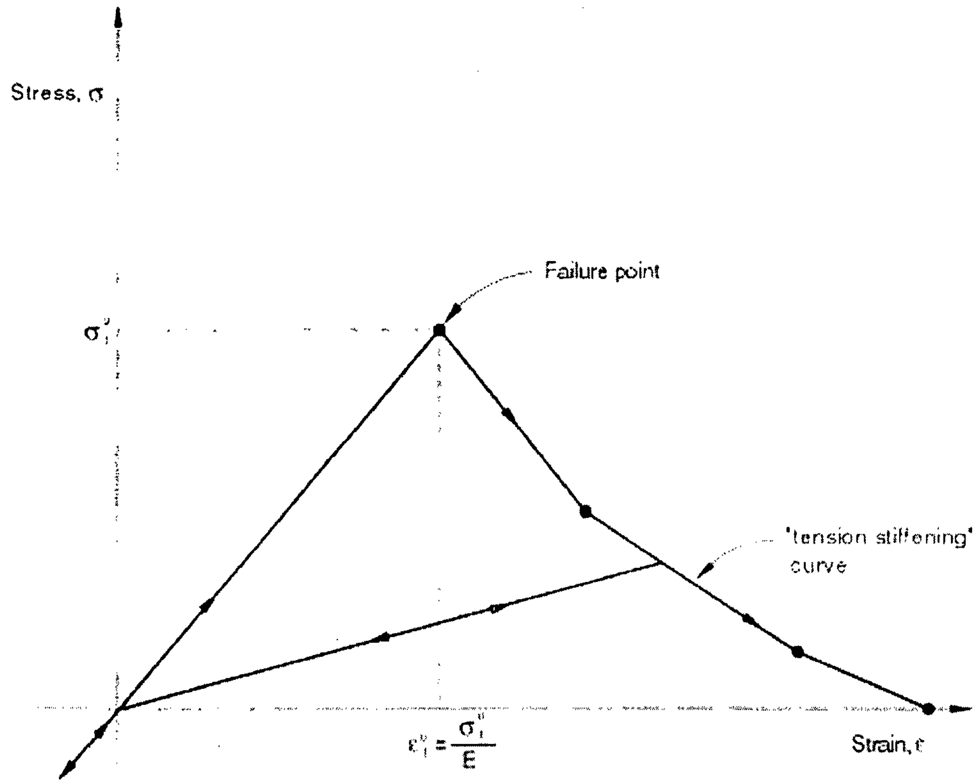
The postfailure behavior for direct straining across cracks is modeled with tension stiffening, which allows you to define the strain-softening behavior for cracked concrete. This behavior also allows for the effects of the reinforcement interaction with concrete to be simulated in a simple manner. Tension stiffening is required in the concrete smeared cracking model. You can specify tension stiffening by means of a postfailure stress-strain relation or by applying a fracture energy cracking criterion.

Postfailure stress-strain relation

Specification of strain softening behavior in reinforced concrete generally means specifying the postfailure stress as a function of strain across the crack. In cases with little or no reinforcement this specification often introduces mesh sensitivity in the analysis results in the sense that the finite element predictions do not converge to a unique solution as the mesh is refined because mesh refinement leads to narrower crack bands. This problem typically occurs if only a few discrete cracks form in the structure, and mesh refinement does not result in formation of additional cracks. If cracks are evenly distributed (either due to the effect of rebar or due to the presence of stabilizing elastic material, as in the case of plate bending), mesh sensitivity is less of a concern.

In practical calculations for reinforced concrete, the mesh is usually such that each element contains rebars. The interaction between the rebars and the concrete tends to reduce the mesh sensitivity, provided that a reasonable amount of tension stiffening is introduced in the concrete model to simulate this interaction (Figure 19.6.1–1).

Figure 19.6.1–1 "Tension stiffening" model.



The tension stiffening effect must be estimated; it depends on such factors as the density of reinforcement, the quality of the bond between the rebar and the concrete, the relative size of the concrete aggregate compared to the rebar diameter, and the mesh. A reasonable starting point for relatively heavily reinforced concrete modeled with a fairly detailed mesh is to assume that the strain softening after failure reduces the stress linearly to zero at a total strain of about 10 times the strain at failure. The strain at failure in standard concretes is typically 10^{-4}, which suggests that tension stiffening that reduces the stress to zero at a total strain of about 10^{-3} is reasonable. This parameter should be calibrated to a particular case.

The choice of tension stiffening parameters is important in Abaqus/Standard since, generally, more tension stiffening makes it easier to obtain numerical solutions. Too little tension stiffening will cause the local cracking failure in the concrete to introduce temporarily unstable behavior in the overall response of the model. Few practical designs exhibit such behavior, so that the presence of this type of response in the analysis model usually indicates that the tension stiffening is unreasonably low.

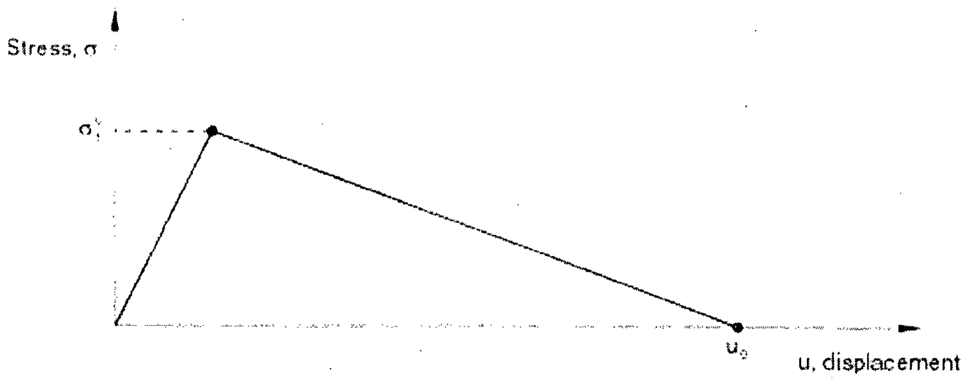
Input File Usage: Use both of the following options:
*CONCRETE
*TENSION STIFFENING, TYPE=STRAIN (default)

Abaqus/CAE Usage: Property module: material editor: **Mechanical→Plasticity→Concrete Smeared Cracking**: **Suboptions→Tension Stiffening**: **Type: Strain**

Fracture energy cracking criterion

As discussed earlier, when there is no reinforcement in significant regions of a concrete model, the strain softening approach for defining tension stiffening may introduce unreasonable mesh sensitivity into the results. Crisfield (1986) discusses this issue and concludes that Hillerborg's (1976) proposal is adequate to allay the concern for many practical purposes. Hillerborg defines the energy required to open a unit area of crack as a material parameter, using brittle fracture concepts. With this approach the concrete's brittle behavior is characterized by a stress-*displacement* response rather than a stress-*strain* response. Under tension a concrete specimen will crack across some section. After it has been pulled apart sufficiently for most of the stress to be removed (so that the elastic strain is small), its length will be determined primarily by the opening at the crack. The opening does not depend on the specimen's length (Figure 19.6.1–2).

Figure 19.6.1–2 Fracture energy cracking model.



Implementation

The implementation of this stress-displacement concept in a finite element model requires the definition of a characteristic length associated with an integration point. The characteristic crack length is based on the element geometry and formulation: it is a typical length of a line across an element for a first-order element; it is half of the same typical length for a second-order element. For beams and trusses it is a characteristic length along the element axis. For membranes and shells it is a characteristic length in the reference surface. For axisymmetric elements it is a characteristic length in the r–z plane only. For cohesive elements it is equal to the constitutive thickness. This definition of the characteristic crack length is used because the direction in which cracks will occur is not known in advance. Therefore, elements with large aspect ratios will have rather different behavior depending on the direction in which they crack: some mesh sensitivity remains because of this effect, and elements that are as close to square as possible are recommended.

This approach to modeling the concrete's brittle response requires the specification of the displacement u_0 at which a linear approximation to the postfailure strain softening gives zero stress (see Figure 19.6.1–2).

The failure stress, σ_t^u, occurs at a failure *strain* (defined by the failure stress divided by the Young's modulus); however, the stress goes to zero at an ultimate *displacement*, u_0, that is independent of the specimen length. The implication is that a displacement-loaded specimen can remain in static equilibrium after failure only if the specimen is short enough so that the strain at failure, ε_t^u, is less than the strain at this value of the displacement:

$$\varepsilon_t^u < u_0/L.$$

where *L* is the length of the specimen.

Input File Usage: Use both of the following options:
*CONCRETE
*TENSION STIFFENING, TYPE=DISPLACEMENT

Abaqus/CAE Usage: Property module: material editor: **Mechanical⟶Plasticity⟶ Concrete Smeared Cracking: Suboptions⟶Tension Stiffening: Type: Displacement**

Obtaining the ultimate displacement

The ultimate displacement, u_0, can be estimated from the fracture energy per unit area, G_f, as $u_0 = 2G_f/\sigma_t^u$, where σ_t^u is the maximum tensile stress that the concrete can carry. Typical values for u_0 are 0.05 mm (2 ı10^{-3} in) for a normal concrete to 0.08 mm (3 ı10^{-3} in) for a high strength concrete. A typical value for ε_t^u is about 10^{-4}, so that the requirement is that $L < 500$ mm (20 in).

Critical length

If the specimen is longer than the critical length, *L*, more strain energy is stored in the specimen than can be dissipated by the cracking process when it cracks under fixed displacement. Some of the strain energy must, therefore, be converted into kinetic energy, and the failure event must be dynamic even under prescribed displacement loading. This implies that, when this approach is used in finite elements, characteristic element dimensions must be less than this critical length, or additional (dynamic) considerations must be included. The analysis input file processor checks the characteristic length of each element using this concrete model and will not allow any element to have a characteristic length that exceeds u_0/ε_t^u. You must remesh with smaller elements where necessary or use the stress-strain definition of tension stiffening. Since the fracture energy approach is generally used only for plain concrete, this rarely places any limit on the meshing.

Cracked shear retention

As the concrete cracks, its shear stiffness is diminished. This effect is defined by specifying the reduction in the shear modulus as a function of the opening strain across the crack. You can also specify a reduced shear modulus for closed cracks. This reduced shear modulus will also have an effect when the normal stress across a crack becomes compressive. The new shear stiffness will have been degraded by the presence of the crack.

The modulus for shearing of cracks is defined as ϱG, where *G* is the elastic shear modulus of the uncracked concrete and ϱ is a multiplying factor. The shear retention model assumes that the shear stiffness of open cracks reduces linearly to zero as the crack opening increases:

$$\varrho = (1 - \varepsilon/\varepsilon^{max}) \quad \text{for} \quad \varepsilon < \varepsilon^{max}, \qquad \varrho = 0 \quad \text{for} \quad \varepsilon \geq \varepsilon^{max}.$$

where ε is the direct strain across the crack and ε^{max} is a user-specified value. The model also assumes that cracks that subsequently close have a reduced shear modulus:

$$\varrho = \varrho^{close} \quad \text{for} \quad \varepsilon < 0.$$

where you specify ϱ^{close}.

ϱ^{close} and ε^{max} can be defined with an optional dependency on temperature and/or predefined field variables. If shear retention is not included in the material definition for the concrete smeared cracking model, Abaqus/Standard will automatically invoke the default behavior for shear retention such that the shear response is unaffected by cracking (full shear retention). This assumption is often reasonable: in many cases, the overall response is not strongly dependent on the amount of shear retention.

Input File Usage: Use both of the following options:
*CONCRETE
*SHEAR RETENTION

Abaqus/CAE Usage: Property module: material editor: **Mechanical→Plasticity→Concrete Smeared Cracking**: **Suboptions→Shear Retention**

Compressive behavior

When the principal stress components are dominantly compressive, the response of the concrete is modeled by an elastic-plastic theory using a simple form of yield surface written in terms of the equivalent pressure stress, *p*, and the Mises equivalent deviatoric stress, *q*; this surface is illustrated in Figure 19.6.1–5. Associated flow and isotropic hardening are used. This model significantly simplifies the actual behavior. The associated flow assumption generally over-predicts the inelastic volume strain. The yield surface cannot be matched accurately to data in triaxial tension and triaxial compression tests because of the omission of third stress invariant dependence. When the concrete is strained beyond the ultimate stress point, the assumption that the elastic response is not affected by the inelastic deformation is not realistic. In addition, when concrete is subjected to very high pressure stress, it exhibits inelastic response: no attempt has been made to build this behavior into the model.

The simplifications associated with compressive behavior are introduced for the sake of computational efficiency. In particular, while the assumption of associated flow is not justified by experimental data, it can provide results that are acceptably close to measurements, provided that the range of pressure stress in the problem is not large. From a computational viewpoint, the associated flow assumption leads to enough symmetry in the Jacobian matrix of the integrated constitutive model (the "material stiffness matrix") such that the overall equilibrium equation solution usually does not require unsymmetric equation solution. All of these limitations could be removed at some sacrifice in computational cost.

You can define the stress-strain behavior of plain concrete in uniaxial compression outside the elastic range. Compressive stress data are provided as a tabular function of plastic strain and, if desired, temperature and field variables. Positive (absolute) values should be given for the compressive stress and strain. The stress-strain curve can be defined beyond the ultimate stress, into the strain-softening regime.

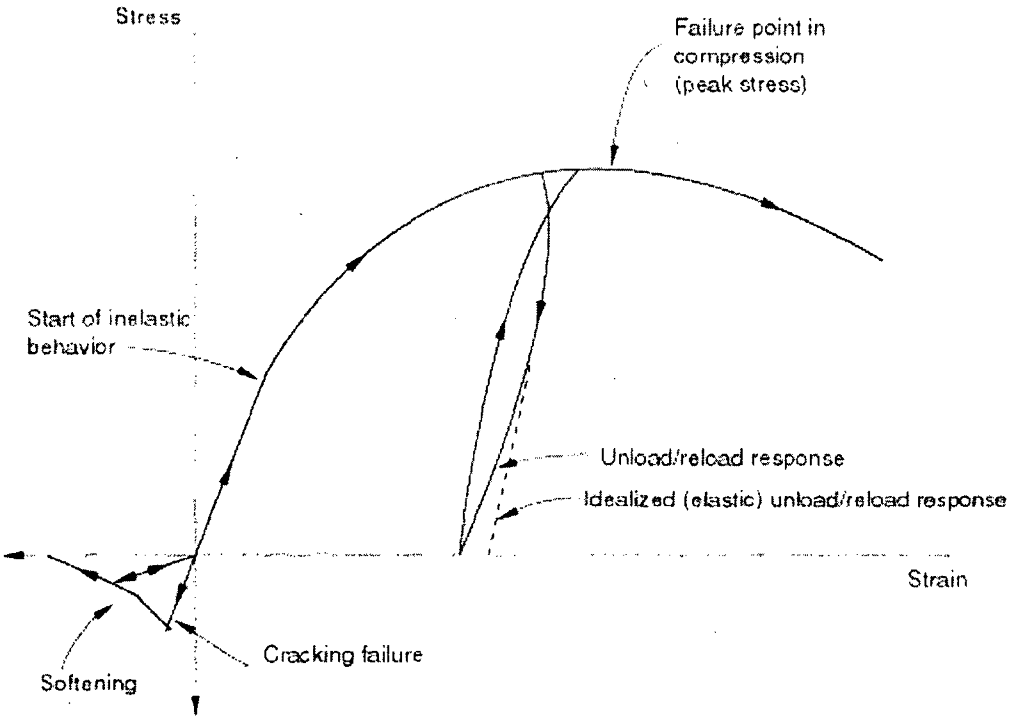
Input File Usage: *CONCRETE

Abaqus/CAE Usage: Property module: material editor: **Mechanical→Plasticity→Concrete Smeared Cracking**

Uniaxial and multiaxial behavior

The cracking and compressive responses of concrete that are incorporated in the concrete model are illustrated by the uniaxial response of a specimen shown in Figure 19.6.1–3.

Figure 19.6.1–3 Uniaxial behavior of plain concrete.



When concrete is loaded in compression, it initially exhibits elastic response. As the stress is increased, some nonrecoverable (inelastic) straining occurs and the response of the material softens. An ultimate stress is reached, after which the material loses strength until it can no longer carry any stress. If the load is removed at some point after inelastic straining has occurred, the unloading response is softer than the initial elastic response: the elasticity has been damaged. This effect is ignored in the model, since we assume that the applications involve primarily monotonic straining, with only occasional, minor unloadings. When a uniaxial concrete specimen is loaded in tension, it responds elastically until, at a stress that is typically 7%–10% of the ultimate compressive stress, cracks form. Cracks form so quickly that, even in the stiffest testing machines available, it is very difficult to observe the actual behavior. The model assumes that cracking causes damage, in the sense that open cracks can be represented by a loss of elastic stiffness. It is also assumed that there is no permanent strain associated with cracking. This will allow cracks to close completely if the stress across them becomes compressive.

In multiaxial stress states these observations are generalized through the concept of surfaces of failure and flow in stress space. These surfaces are fitted to experimental data. The surfaces used are shown in Figure 19.6.1–4 and Figure 19.6.1–5.

Figure 19.6.1–4 Yield and failure surfaces in plane stress.

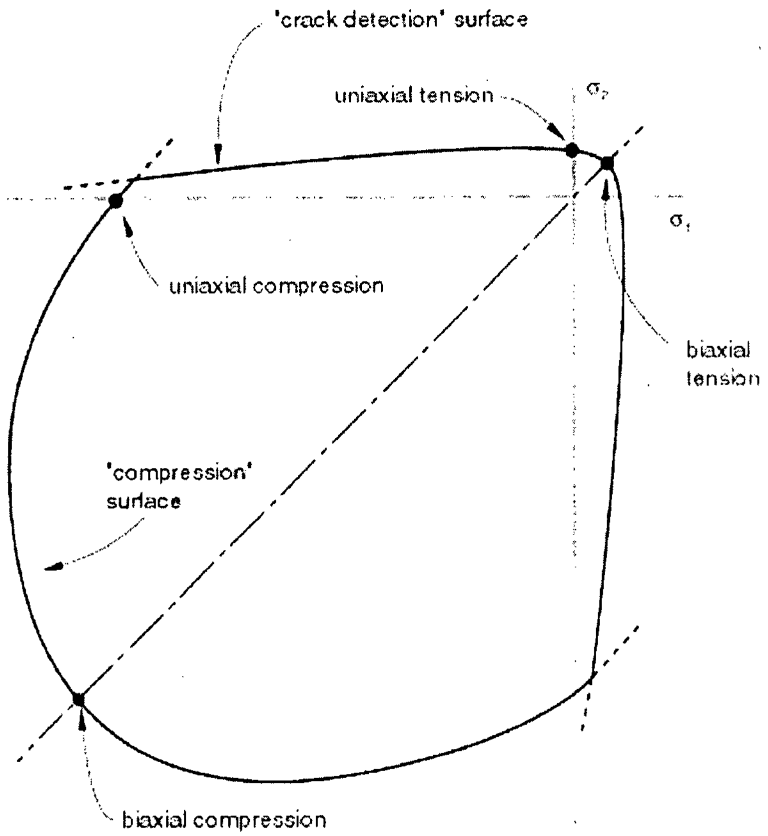
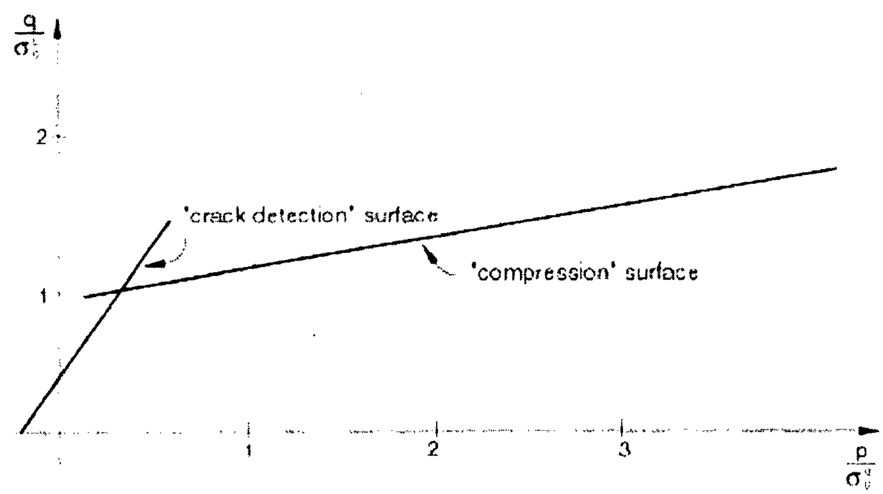


Figure 19.6.1–5 Yield and failure surfaces in the (p–q) plane.



Failure surface

You can specify failure ratios to define the shape of the failure surface (possibly as a function of temperature and predefined field variables). Four failure ratios can be specified:

- The ratio of the ultimate biaxial compressive stress to the ultimate uniaxial compressive stress.
- The absolute value of the ratio of the uniaxial tensile stress at failure to the ultimate uniaxial compressive stress.
- The ratio of the magnitude of a principal component of plastic strain at ultimate stress in biaxial compression to the plastic strain at ultimate stress in uniaxial compression.
- The ratio of the tensile principal stress at cracking, in plane stress, when the other principal stress is at the ultimate compressive value, to the tensile cracking stress under uniaxial tension.

Default values of the above ratios are used if you do not specify them.

Input File Usage: *FAILURE RATIOS

Abaqus/CAE Usage: Property module: material editor: **Mechanical→Plasticity→Concrete Smeared Cracking**: **Suboptions→Failure Ratios**

Response to strain reversals

Because the model is intended for application to problems involving relatively monotonic straining, no attempt is made to include prediction of cyclic response or of the reduction in the elastic stiffness caused by inelastic straining under predominantly compressive stress. Nevertheless, it is likely that, even in those applications for which the model is designed, the strain trajectories will not be entirely radial, so that the model should predict the response to occasional strain reversals and strain trajectory direction changes in a reasonable way. Isotropic hardening of the "compressive" yield surface forms the basis of this aspect of the model's inelastic response prediction when the principal stresses are dominantly compressive.

Calibration

A minimum of two experiments, uniaxial compression and uniaxial tension, is required to calibrate the simplest version of the concrete model (using all possible defaults and assuming temperature and field variable independence). Other experiments may be required to gain accuracy in postfailure behavior.

Uniaxial compression and tension tests

The uniaxial compression test involves compressing the sample between two rigid platens. The load and displacement in the direction of loading are recorded. From this, you can extract the stress-strain curve required for the concrete model directly. The uniaxial tension test is much more difficult to perform in the sense that it is necessary to have a stiff testing machine to be able to record the postfailure response. Quite often this test is not available, and you make an assumption about the tensile failure strength of the concrete (usually about 7%–10% of the compressive strength). The choice of tensile cracking stress is important; numerical problems may arise if very low cracking stresses are used (less than 1/100 or 1/1000 of the compressive strength).

Postcracking tensile behavior

The calibration of the postfailure response depends on the reinforcement present in the concrete. For plain concrete simulations the stress-displacement tension stiffening model should be used. Typical values for u_0 are 0.05 mm (2×10^{-3} in) for a normal concrete to 0.08 mm (3×10^{-3} in) for a high-strength concrete. For reinforced concrete simulations the stress-strain tension stiffening model should be used. A reasonable starting point for relatively heavily reinforced concrete modeled with a fairly detailed mesh is to assume that the strain softening after failure reduces the stress linearly to zero at a total strain of about 10 times the strain at failure. Since the strain at failure in standard concretes is typically 10^{-4}, this suggests that tension stiffening that reduces the stress to zero at a total strain of about 10^{-3} is reasonable. This parameter should be calibrated to a particular case.

Postcracking shear behavior

Combined tension and shear experiments are used to calibrate the postcracking shear behavior in Abaqus/Standard. These experiments are quite difficult to perform. If the test data are not available, a reasonable starting point is to assume that the shear retention factor, ρ, goes linearly to zero at the same crack opening strain used for the tension stiffening model.

Biaxial yield and flow parameters

Biaxial experiments are required to calibrate the biaxial yield and flow parameters used to specify the failure ratios. If these are not available, the defaults can be used.

Temperature dependence

The calibration of temperature dependence requires the repetition of all the above experiments over the range of interest.

Comparison with experimental results

With proper calibration, the concrete model should produce reasonable results for mostly monotonic loadings. Comparison of the predictions of the model with the experimental results of Kupfer and Gerstle (1973) are shown in Figure 19.6.1–6 and Figure 19.6.1–7.

Figure 19.6.1–6 Comparison of model prediction and Kupfer and Gerstle's data for a uniaxial compression test.

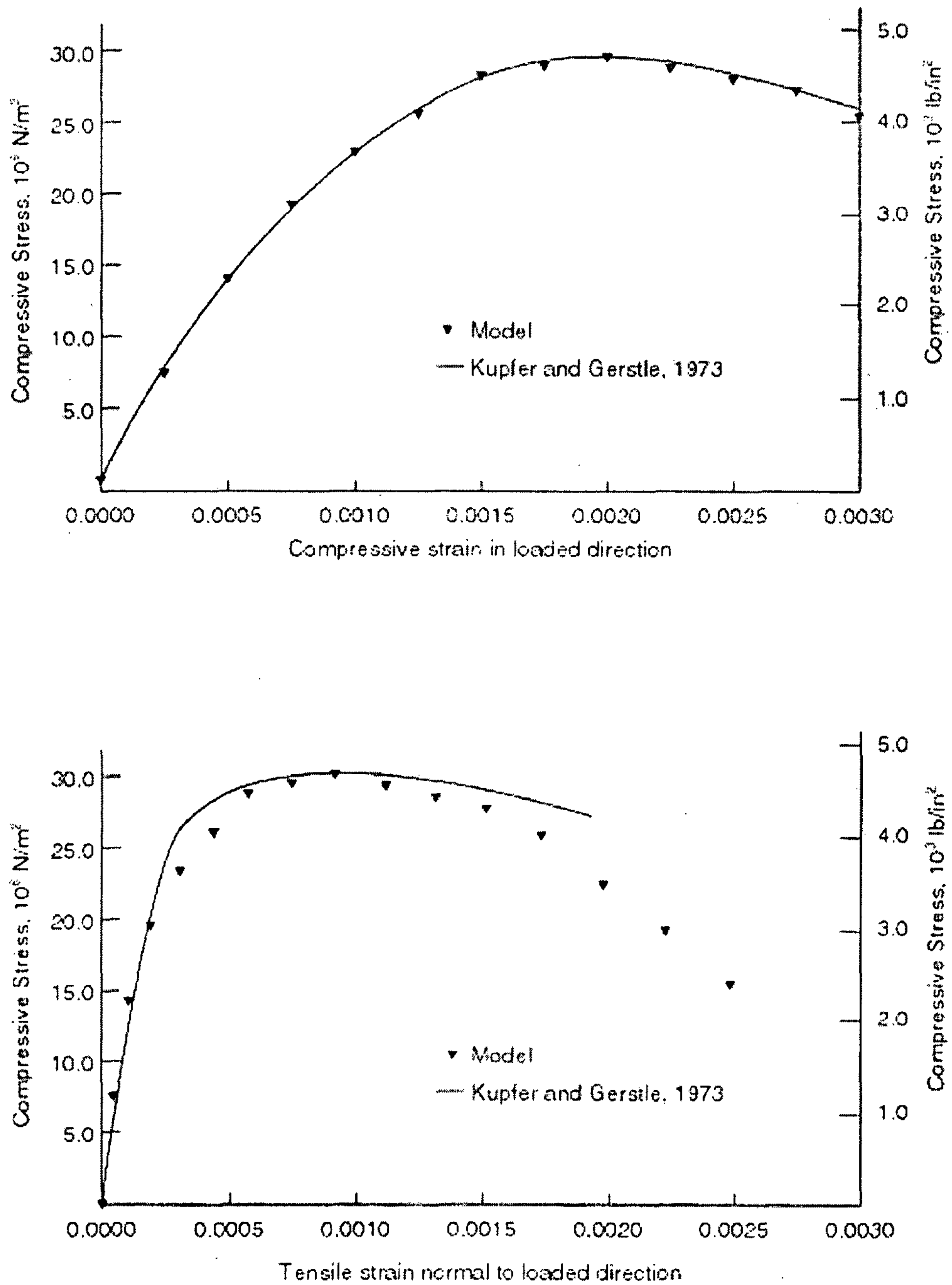
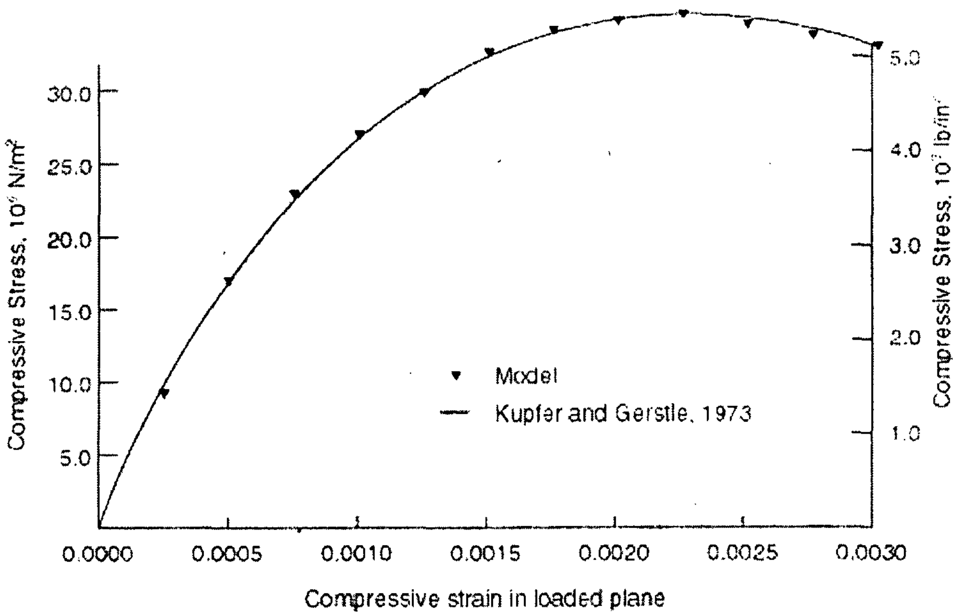
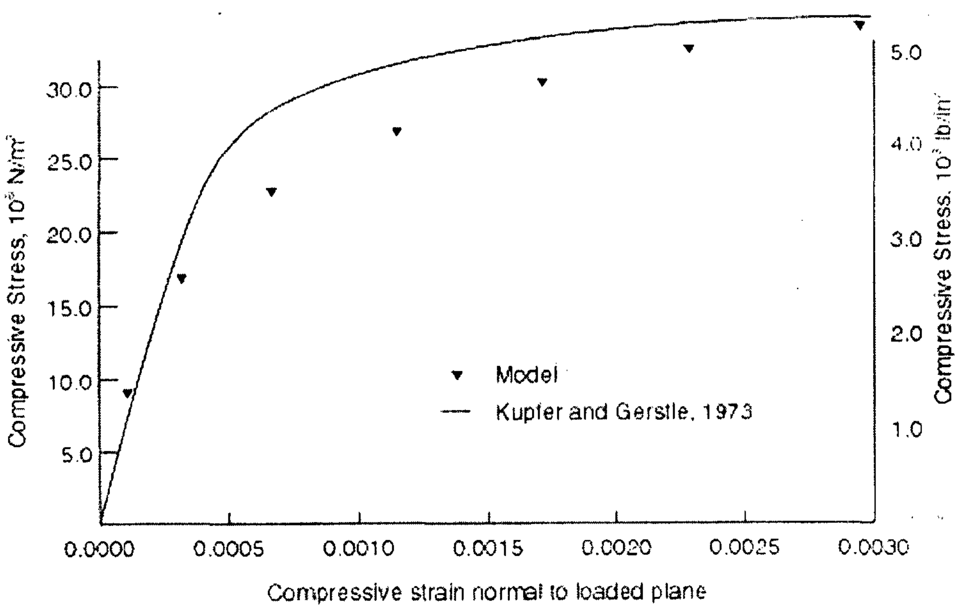


Figure 19.6.1–7 Comparison of model prediction and Kupfer and Gerstle's data for a biaxial compression test.





Elements

Abaqus/Standard offers a variety of elements for use with the smeared crack concrete model: beam, shell, plane stress, plane strain, generalized plane strain, axisymmetric, and three-dimensional elements.

For general shell analysis more than the default number of five integration points through the thickness of the shell should be used; nine thickness integration points are commonly used to model progressive failure of the concrete through the thickness with acceptable accuracy.

Output

In addition to the standard output identifiers available in Abaqus/Standard ("Abaqus/Standard output variable identifiers," Section 4.2.1), the following variables relate specifically to material points in the smeared crack concrete model:

CRACK	Unit normal to cracks in concrete.
CONF	Number of cracks at a concrete material point.

Additional references

- Crisfield, M. A., "Snap-Through and Snap-Back Response in Concrete Structures and the Dangers of Under-Integration," International Journal for Numerical Methods in Engineering, vol. 22, pp. 751–767, 1986.
- Hillerborg, A., M. Modeer, and P. E. Petersson, "Analysis of Crack Formation and Crack Growth in Concrete by Means of Fracture Mechanics and Finite Elements," Cement and Concrete Research, vol. 6, pp. 773–782, 1976.
- Kupfer, H. B., and K. H. Gerstle, "Behavior of Concrete under Biaxial Stresses," Journal of Engineering Mechanics Division, ASCE, vol. 99 853, 1973.

19.6.2 Cracking model for concrete

Products: Abaqus/Explicit Abaqus/CAE

References

- "Material library: overview," Section 17.1.1
- "Inelastic behavior," Section 19.1.1
- *BRITTLE CRACKING
- *BRITTLE FAILURE
- *BRITTLE SHEAR
- "Defining brittle cracking" in "Defining other mechanical models," Section 12.9.4 of the Abaqus/CAE User's Manual

Overview

The brittle cracking model in Abaqus/Explicit:

- provides a capability for modeling concrete in all types of structures: beams, trusses, shells and solids;
- can also be useful for modeling other materials such as ceramics or brittle rocks;
- is designed for applications in which the behavior is dominated by tensile cracking;
- assumes that the compressive behavior is always linear elastic;
- must be used with the linear elastic material model ("Linear elastic behavior," Section 18.2.1), which also defines the material behavior completely prior to cracking;
- is most accurate in applications where the brittle behavior dominates such that the assumption that the material is linear elastic in compression is adequate;
- can be used for plain concrete, even though it is intended primarily for the analysis of reinforced concrete structures;
- allows removal of elements based on a brittle failure criterion; and
- is defined in detail in "A cracking model for concrete and other brittle materials," Section 4.5.3 of the Abaqus Theory Manual.

See "Inelastic behavior," Section 19.1.1, for a discussion of the concrete models available in Abaqus.

Reinforcement

Reinforcement in concrete structures is typically provided by means of rebars. Rebars are one-dimensional strain theory elements (rods) that can be defined singly or embedded in oriented surfaces. Rebars are discussed in "Defining rebar as an element property," Section 2.2.4. They are typically used with elastic-plastic material behavior and are superposed on a mesh of standard element types used to model the plain concrete. With this modeling approach, the concrete cracking behavior is considered independently of the rebar. Effects associated with the rebar/concrete interface, such as bond slip and dowel action, are modeled approximately by introducing some "tension stiffening" into the concrete cracking model to simulate load transfer across cracks through the rebar.

Cracking

Abaqus/Explicit uses a smeared crack model to represent the discontinuous brittle behavior in concrete. It does not track individual "macro" cracks: instead, constitutive calculations are performed independently at each material point of the finite element model. The presence of cracks enters into these calculations by the way in which the cracks affect the stress and material stiffness associated with the material point.

For simplicity of discussion in this section, the term "crack" is used to mean a direction in which cracking has been detected at the single material calculation point in question: the closest physical concept is that there exists a continuum of micro-cracks in the neighborhood of the point, oriented as determined by the model. The anisotropy introduced by cracking is assumed to be important in the simulations for which the model is intended.

Crack directions

The Abaqus/Explicit cracking model assumes fixed, orthogonal cracks, with the maximum number of cracks at a material point limited by the number of direct stress components present at that material point of the finite element model (a maximum of three cracks in three-dimensional, plane strain, and axisymmetric problems; two cracks in plane stress and shell problems; and one crack in beam or truss problems). Internally, once cracks exist at a point, the component forms of all vector- and tensor-valued quantities are rotated so that they lie in the local system defined by the crack orientation vectors (the normals to the crack faces). The model ensures that these crack face normal vectors will be orthogonal, so that this local crack system is rectangular Cartesian. For output purposes you are offered results of stresses and strains in the global and/or local crack systems.

Crack detection

A simple Rankine criterion is used to detect crack initiation. This criterion states that a crack forms when the maximum principal tensile stress exceeds the tensile strength of the brittle material. Although crack detection is based purely on Mode I fracture considerations, ensuing cracked behavior includes both Mode I (tension softening/stiffening) and Mode II (shear softening/retention) behavior, as described later.

As soon as the Rankine criterion for crack formation has been met, we assume that a first crack has formed. The crack surface is taken to be normal to the direction of the maximum tensile principal stress. Subsequent cracks may form with crack surface normals in the direction of maximum principal tensile stress that is orthogonal to the directions of any existing crack surface normals at the same point.

Cracking is irrecoverable in the sense that, once a crack has occurred at a point, it remains throughout the rest of the calculation. However, crack closing and reopening may take place along the directions of the crack surface normals. The model neglects any permanent strain associated with cracking; that is, it is assumed that the cracks can close completely when the stress across them becomes compressive.

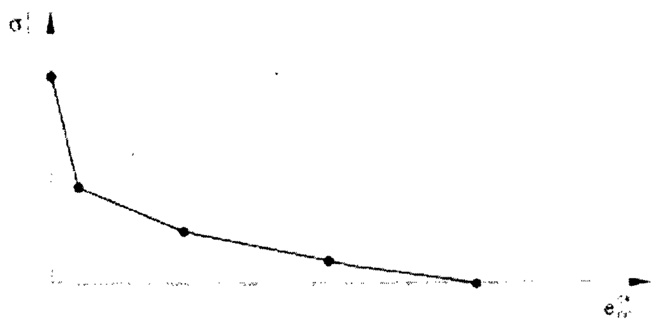
Tension stiffening

You can specify the postfailure behavior for direct straining across cracks by means of a postfailure stress-strain relation or by applying a fracture energy cracking criterion.

Postfailure stress-strain relation

In reinforced concrete the specification of postfailure behavior generally means giving the postfailure stress as a function of strain across the crack (Figure 19.6.2–1). In cases with little or no reinforcement, this introduces mesh sensitivity in the results, in the sense that the finite element predictions do not converge to a unique solution as the mesh is refined because mesh refinement leads to narrower crack bands.

Figure 19.6.2–1 Postfailure stress-strain curve.



In practical calculations for reinforced concrete, the mesh is usually such that each element contains rebars. In this case the interaction between the rebars and the concrete tends to mitigate this effect, provided that a reasonable amount of "tension stiffening" is introduced in the cracking model to simulate this interaction. This requires an estimate of the tension stiffening effect, which depends on factors such as the density of reinforcement, the quality of the bond between the rebar and the concrete, the relative size of the concrete aggregate compared to the rebar diameter, and the mesh. A reasonable starting point for relatively heavily reinforced concrete modeled with a fairly detailed mesh is to assume that the strain softening after failure reduces the stress linearly to zero at a total strain about ten times the strain at failure. Since the strain at failure in standard concretes is typically 10^{-4}, this suggests that tension stiffening that reduces the stress to zero at a total strain of about 10^{-3} is reasonable. This parameter should be calibrated to each particular case. In static applications too little tension stiffening will cause the local cracking failure in the concrete to introduce temporarily unstable behavior in the overall response of the model. Few practical designs exhibit such behavior, so that the presence of this type of response in the analysis model usually indicates that the tension stiffening is unreasonably low.

Input File Usage: *BRITTLE CRACKING, TYPE=STRAIN

Abaqus/CAE Usage: Property module: material editor: **Mechanical→Brittle Cracking: Type: Strain**

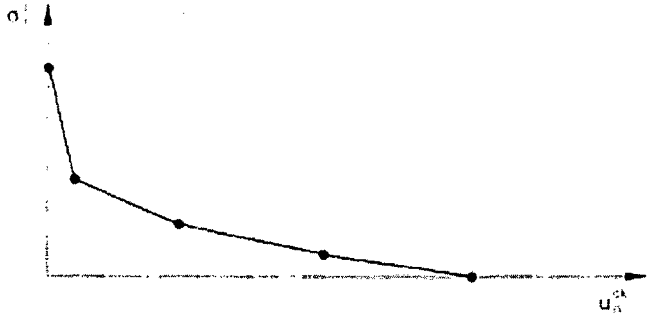
Fracture energy cracking criterion

When there is no reinforcement in significant regions of the model, the tension stiffening approach described above will introduce unreasonable mesh sensitivity into the results. However, it is generally accepted that Hillerborg's (1976) fracture energy proposal is adequate to allay the concern for many practical purposes. Hillerborg defines the energy required to open a unit area of crack in Mode I (G^I_f) as a material parameter, using brittle fracture concepts. With this approach the concrete's brittle behavior is characterized by a stress-*displacement* response rather than a stress-*strain* response. Under tension a concrete specimen will crack across some section; and its length, after it has been pulled apart sufficiently for most of the stress to be removed (so that the elastic strain is small), will be determined primarily by the opening at the crack, which does not depend on the specimen's length.

Implementation

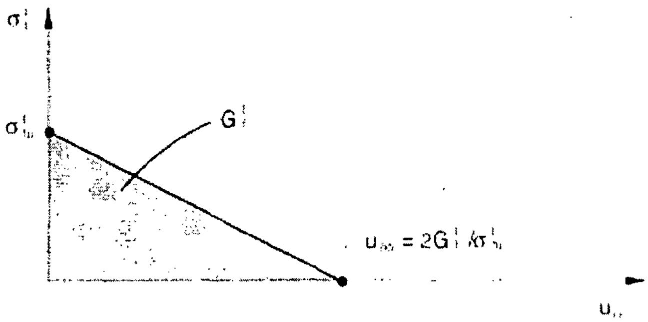
In Abaqus/Explicit this fracture energy cracking model can be invoked by specifying the postfailure stress as a tabular function of displacement across the crack, as illustrated in Figure 19.6.2–2.

Figure 19.6.2–2 Postfailure stress-displacement curve.



Alternatively, the Mode I fracture energy, G_f^I, can be specified directly as a material property; in this case, define the failure stress, σ_{tu}^I, as a tabular function of the associated Mode I fracture energy. This model assumes a linear loss of strength after cracking (Figure 19.6.2–3).

Figure 19.6.2–3 Postfailure stress-fracture energy curve.



The crack normal displacement at which complete loss of strength takes place is, therefore, $u_{n0} = 2G_f^I/\sigma_{tu}^I$. Typical values of G_f^I range from 40 N/m (0.22 lb/in) for a typical construction concrete (with a compressive strength of approximately 20 MPa, 2850 lb/in^2) to 120 N/m (0.67 lb/in) for a high-strength concrete (with a compressive strength of approximately 40 MPa, 5700 lb/in^2).

Input File Usage: Use the following option to specify the postfailure stress as a tabular function of displacement:

*BRITTLE CRACKING, TYPE=DISPLACEMENT

Use the following option to specify the postfailure stress as a tabular function of the fracture energy:

*BRITTLE CRACKING, TYPE=GFI

Abaqus/CAE Usage: Property module: material editor: **Mechanical⟶Brittle Cracking**: **Type: Displacement** or **GFI**

Characteristic crack length

The implementation of the stress-displacement concept in a finite element model requires the definition of a characteristic length associated with a material point. The characteristic crack length is based on the element geometry and formulation: it is a typical length of a line across an element for a first-order element; it is half of the same typical length for a second-order element. For beams and trusses it is a characteristic length along the element axis. For membranes and shells it is a characteristic length in the reference surface. For axisymmetric elements it is a characteristic length in the *r–z* plane only. For cohesive elements it is equal to the constitutive thickness. We use this definition of the characteristic crack length because the direction in which cracks will occur is not known in advance. Therefore, elements with large aspect ratios will have rather different behavior depending on the direction in which they crack: some mesh sensitivity remains because of this effect. Elements that are as close to square as possible are, therefore, recommended unless you can predict the direction in which cracks will form.

Shear retention model

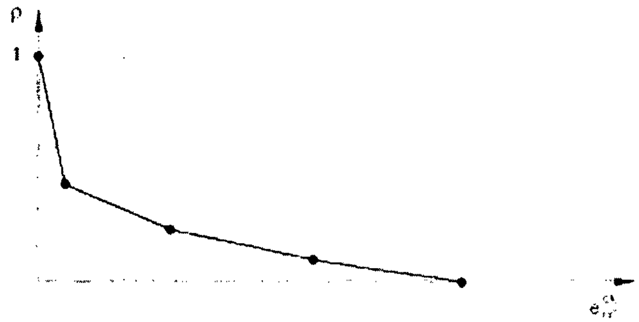
An important feature of the cracking model is that, whereas crack initiation is based on Mode I fracture only, postcracked behavior includes Mode II as well as Mode I. The Mode II shear behavior is based on the common observation that the shear behavior depends on the amount of crack opening. More specifically, the cracked shear modulus is reduced as the crack opens. Therefore, Abaqus/Explicit offers a shear retention model in which the postcracked shear stiffness is defined as a function of the opening strain across the crack; the shear retention model must be defined in the cracking model, and zero shear retention should not be used.

In these models the dependence is defined by expressing the postcracking shear modulus, G_c, as a fraction of the uncracked shear modulus:

$$G_c = \rho(e^{ck}_{nn})\, G,$$

where *G* is the shear modulus of the uncracked material and the shear retention factor, $\rho(e^{ck}_{nn})$, depends on the crack opening strain, e^{ck}_{nn}. You can specify this dependence in piecewise linear form, as shown in Figure 19.6.2–4.

Figure 19.6.2–4 Piecewise linear form of the shear retention model.

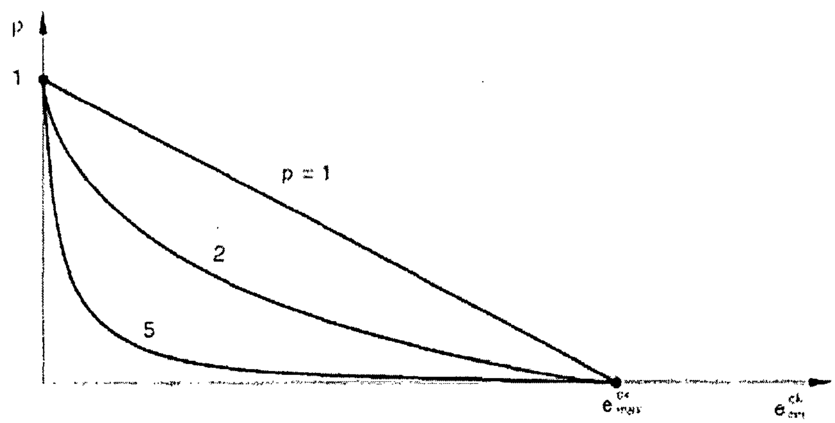


Alternatively, shear retention can be defined in the power law form:

$$\rho(e_{nn}^{ck}) = \left(1 - \frac{e_{nn}^{ck}}{e_{max}^{ck}}\right)^{p}.$$

where *p* and e_{max}^{ck} are material parameters. This form, shown in Figure 19.6.2–5, satisfies the requirements that $\rho \rightarrow 1$ as $e_{nn}^{ck} \rightarrow 0$ (corresponding to the state before crack initiation) and $\rho \rightarrow 0$ as $e_{nn}^{ck} \rightarrow e_{max}^{ck}$ (corresponding to complete loss of aggregate interlock). See "A cracking model for concrete and other brittle materials," Section 4.5.3 of the Abaqus Theory Manual, for a discussion of how shear retention is calculated in the case of two or more cracks.

Figure 19.6.2–5 Power law form of the shear retention model.



Input File Usage: Use the following option to specify the piecewise linear form of the shear retention model:
*BRITTLE SHEAR, TYPE=RETENTION FACTOR

Use the following option to specify the power law form of the shear retention model:

*BRITTLE SHEAR, TYPE=POWER LAW

Abaqus/CAE Usage: Property module: material editor: **Mechanical→Brittle Cracking: Suboptions→Brittle Shear Type: Retention Factor** or **Power Law**

Calibration

One experiment, a uniaxial tension test, is required to calibrate the simplest version of the brittle cracking model. Other experiments may be required to gain accuracy in postfailure behavior.

Uniaxial tension test

This test is difficult to perform because it is necessary to have a very stiff testing machine to record the postcracking response. Quite often such equipment is not available; in this situation you must make an assumption about the tensile failure strength of the material and the postcracking response. For concrete the assumption usually made is that the tensile strength is 7–10% of the compressive strength. Uniaxial compression tests can be performed much more easily, so the compressive strength of concrete is usually known.

Postcracking tensile behavior

The values given for tension stiffening are a very important aspect of simulations using the Abaqus/Explicit brittle cracking model. The postcracking tensile response is highly dependent on the reinforcement present in the concrete. In simulations of unreinforced concrete, the tension stiffening models that are based on fracture energy concepts should be utilized. If reliable experimental data are not available, typical values that can be used were discussed before: common values of G_f^I range from 40 N/m (0.22 lb/in) for a typical construction concrete (with a compressive strength of approximately 20 MPa, 2850 lb/in^2) to 120 N/m (0.67 lb/in) for a high-strength concrete (with a compressive strength of approximately 40 MPa, 5700 lb/in^2). In simulations of reinforced concrete the stress-strain tension stiffening model should be used; the amount of tension stiffening depends on the reinforcement present, as discussed before. A reasonable starting point for relatively heavily reinforced concrete modeled with a fairly detailed mesh is to assume that the strain softening after failure reduces the stress linearly to zero at a total strain about ten times the strain at failure. Since the strain at failure in standard concretes is typically 10^{-4}, this suggests that tension stiffening that reduces the stress to zero at a total strain of about 10^{-3} is reasonable. This parameter should be calibrated to each particular case.

Postcracking shear behavior

Calibration of the postcracking shear behavior requires combined tension and shear experiments, which are difficult to perform. If such test data are not available, a reasonable starting point is to assume that the shear retention factor, ρ, goes linearly to zero at the same crack opening strain used for the tension stiffening model.

Brittle failure criterion

You can define brittle failure of the material. When one, two, or all three local direct cracking strain (displacement) components at a material point reach the value defined as the failure strain (displacement), the material point fails and all the stress components are set to zero. If all of the material points in an element fail, the element is removed from the mesh. For example, removal of a first-order reduced-integration solid element takes place as soon as its only integration point fails. However, all through-the-thickness integration points must fail before a shell element is removed from the mesh.

If the postfailure relation is defined in terms of stress versus strain, the failure strain must be given as the failure criterion. If the postfailure relation is defined in terms of stress versus displacement or stress versus fracture energy, the failure displacement must be given as the failure criterion. The failure strain (displacement) can be specified as a function of temperature and/or predefined field variables.

You can control how many cracks at a material point must fail before the material point is considered to have failed; the default is one crack. The number of cracks that must fail can only be one for beam and truss elements; it cannot be greater than two for plane stress and shell elements; and it cannot be greater than three otherwise.

Input File Usage: *BRITTLE FAILURE, CRACKS=*n*

Abaqus/CAE Usage: Property module: material editor: **Mechanical→Brittle Cracking**: **Suboptions→Brittle Failure** and select **Failure Criteria**: **Unidirectional**, **Bidirectional**, or **Tridirectional** to indicate the number of cracks that must fail for the material point to fail.

Determining when to use the brittle failure criterion

The brittle failure criterion is a crude way of modeling failure in Abaqus/Explicit and should be used with care. The main motivation for including this capability is to help in computations where not removing an element that can no longer carry stress may lead to excessive distortion of that element and subsequent premature termination of the simulation. For example, in a monotonically loaded structure whose failure mechanism is expected to be dominated by a single tensile macrofracture (Mode I cracking), it may be reasonable to use the brittle failure criterion to remove elements. On the other hand, the fact that the brittle material loses its ability to carry tensile stress does not preclude it from withstanding compressive stress; therefore, it may not be appropriate to remove elements if the material is expected to carry compressive loads after it has failed in tension. An example may be a shear wall subjected to cyclic loading as a result of some earthquake excitation; in this case cracks that develop completely under tensile stress will be able to carry compressive stress when load reversal takes place.

Thus, the effective use of the brittle failure criterion relies on you having some knowledge of the structural behavior and potential failure mechanism. The use of the brittle failure criterion based on an incorrect user assumption of the failure mechanism will generally result in an incorrect simulation.

Selecting the number of cracks that must fail before the material point is considered to have failed

When you define brittle failure, you can control how many cracks must open to beyond the failure value before a material point is considered to have failed. The default number of cracks (one) should be used for most structural applications where failure is dominated by Mode I type cracking. However, there are cases in which you should specify a higher number because multiple cracks need to form to develop the eventual failure mechanism. One example may be an unreinforced, deep concrete beam where the failure mechanism is dominated by shear; in this case it is possible that two cracks need to form at each material point for the shear failure mechanism to develop.

Again, the appropriate choice of the number of cracks that must fail relies on your knowledge of the structural and failure behaviors.

Using brittle failure with rebar

It is possible to use the brittle failure criterion in brittle cracking elements for which rebar are also defined; the obvious application is the modeling of reinforced concrete. When such elements fail according to the brittle failure criterion, the brittle cracking contribution to the element stress carrying capacity is removed but the rebar contribution to the element stress carrying capacity is not removed. However, if you also include shear failure in the rebar material definition, the rebar contribution to the element stress carrying capacity will also be removed if the shear failure criterion specified for the rebar is satisfied. This allows the modeling of progressive failure of an under-reinforced concrete structure where the concrete fails first followed by ductile failure of the reinforcement.

Elements

Abaqus/Explicit offers a variety of elements for use with the cracking model: truss; shell; two-dimensional beam; and plane stress, plane strain, axisymmetric, and three-dimensional continuum elements. The model cannot be used with three-dimensional beam elements. Plane triangular, triangular prism, and tetrahedral elements are not recommended for use in reinforced concrete analysis since these elements do not support the use of rebar.

Output

In addition to the standard output identifiers available in Abaqus/Explicit (see "Abaqus/Explicit output variable identifiers," Section 4.2.2), the following output variables relate directly to material points that use the brittle cracking model:

CKE	All cracking strain components.
CKLE	All cracking strain components in local crack axes.
CKEMAG	Cracking strain magnitude.
CKLS	All stress components in local crack axes.
CRACK	Crack orientations.
CKSTAT	Crack status of each crack.
STATUS	Status of element (brittle failure model). The status of an element is 1.0 if the element is active and 0.0 if the element is not.

Additional reference

- Hillerborg, A., M. Modeer, and P. E. Petersson, "Analysis of Crack Formation and Crack Growth in Concrete by Means of Fracture Mechanics and Finite Elements," Cement and Concrete Research, vol. 6, pp. 773–782, 1976.

19.6.3 Concrete damaged plasticity

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

References

- "Material library: overview," Section 17.1.1
- "Inelastic behavior," Section 19.1.1
- *CONCRETE DAMAGED PLASTICITY
- *CONCRETE TENSION STIFFENING
- *CONCRETE COMPRESSION HARDENING
- *CONCRETE TENSION DAMAGE
- *CONCRETE COMPRESSION DAMAGE
- "Defining concrete damaged plasticity" in "Defining plasticity," Section 12.9.2 of the Abaqus/CAE User's Manual

Overview

The concrete damaged plasticity model in Abaqus:

- provides a general capability for modeling concrete and other quasi-brittle materials in all types of structures (beams, trusses, shells, and solids);
- uses concepts of isotropic damaged elasticity in combination with isotropic tensile and compressive plasticity to represent the inelastic behavior of concrete;
- can be used for plain concrete, even though it is intended primarily for the analysis of reinforced concrete structures;
- can be used with rebar to model concrete reinforcement;
- is designed for applications in which concrete is subjected to monotonic, cyclic, and/or dynamic loading under low confining pressures;
- consists of the combination of nonassociated multi-hardening plasticity and scalar (isotropic) damaged elasticity to describe the irreversible damage that occurs during the fracturing process;
- allows user control of stiffness recovery effects during cyclic load reversals;
- can be defined to be sensitive to the rate of straining;
- can be used in conjunction with a viscoplastic regularization of the constitutive equations in Abaqus/Standard to improve the convergence rate in the softening regime;
- requires that the elastic behavior of the material be isotropic and linear (see "Defining isotropic elasticity" in "Linear elastic behavior," Section 18.2.1); and
- is defined in detail in "Damaged plasticity model for concrete and other quasi-brittle materials," Section 4.5.2 of the Abaqus Theory Manual.

See "Inelastic behavior," Section 19.1.1, for a discussion of the concrete models available in Abaqus.

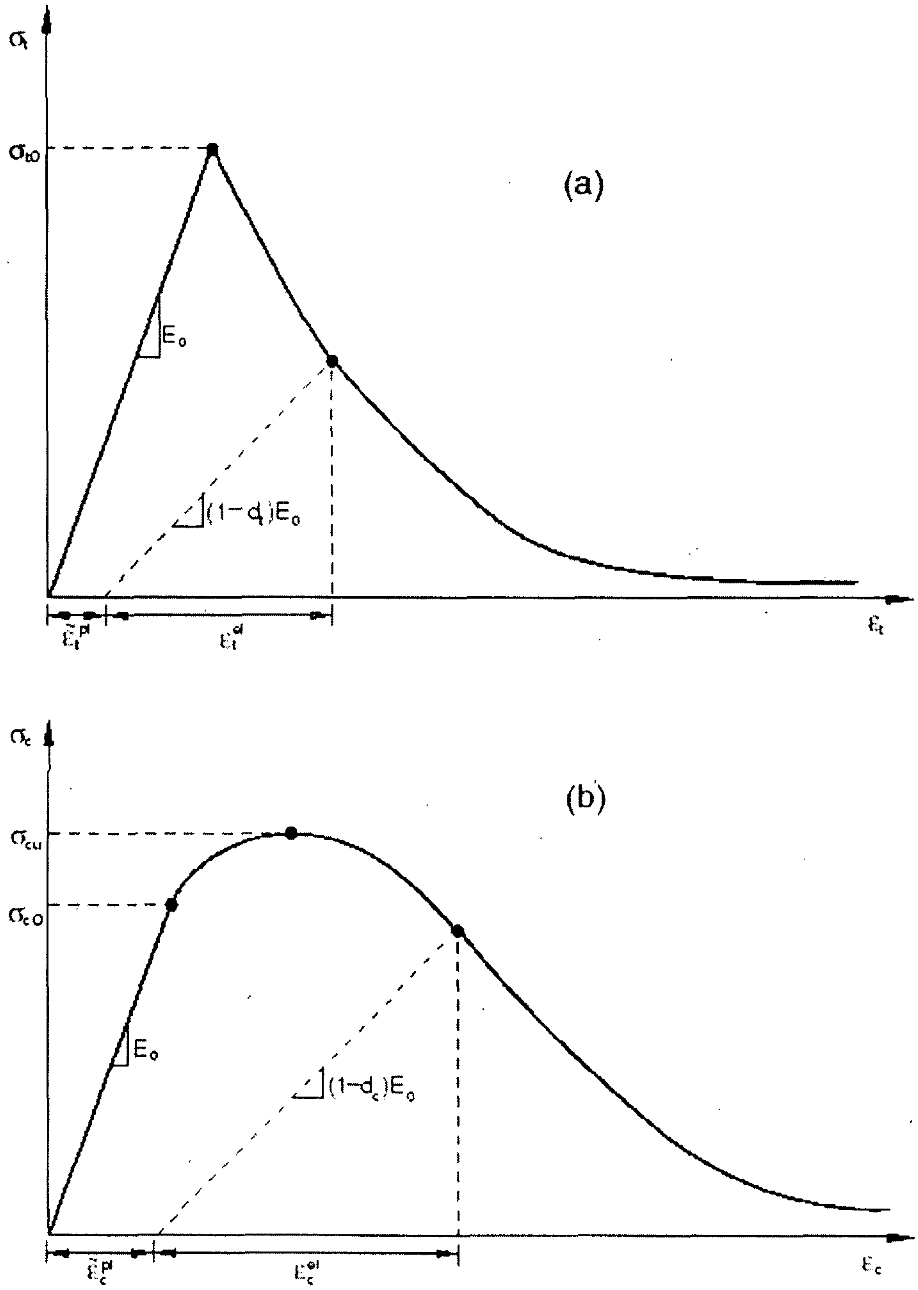
Mechanical behavior

The model is a continuum, plasticity-based, damage model for concrete. It assumes that the main two failure mechanisms are tensile cracking and compressive crushing of the concrete material. The evolution of the yield (or failure) surface is controlled by two hardening variables, $\tilde{\varepsilon}_t^{pl}$ and $\tilde{\varepsilon}_c^{pl}$, linked to failure mechanisms under tension and compression loading, respectively. We refer to $\tilde{\varepsilon}_t^{pl}$ and $\tilde{\varepsilon}_c^{pl}$ as tensile and compressive equivalent plastic strains, respectively. The following sections discuss the main assumptions about the mechanical behavior of concrete.

Uniaxial tension and compression stress behavior

The model assumes that the uniaxial tensile and compressive response of concrete is characterized by damaged plasticity, as shown in Figure 19.6.3–1.

Figure 19.6.3–1 Response of concrete to uniaxial loading in tension (a) and compression (b).



Under uniaxial tension the stress-strain response follows a linear elastic relationship until the value of the failure stress, σ_{t0}, is reached. The failure stress corresponds to the onset of micro-cracking in the concrete material. Beyond the failure stress the formation of micro-cracks is represented macroscopically with a softening stress-strain response, which induces strain localization in the concrete structure. Under uniaxial compression the response is linear until the value of initial yield, σ_{c0}. In the plastic regime the response is typically characterized by stress hardening followed by strain softening beyond the ultimate stress, σ_{cu}. This representation, although somewhat simplified, captures the main features of the response of concrete.

It is assumed that the uniaxial stress-strain curves can be converted into stress versus plastic-strain curves. (This conversion is performed automatically by Abaqus from the user-provided stress versus "inelastic" strain data, as explained below.) Thus,

$$\sigma_t = \sigma_t(\tilde{\varepsilon}_t^{pl}, \dot{\tilde{\varepsilon}}_t^{pl}, \theta, f_i),$$
$$\sigma_c = \sigma_c(\tilde{\varepsilon}_c^{pl}, \dot{\tilde{\varepsilon}}_c^{pl}, \theta, f_i),$$

where the subscripts t and c refer to tension and compression, respectively; $\tilde{\varepsilon}_t^{pl}$ and $\tilde{\varepsilon}_c^{pl}$ are the equivalent plastic strains, $\dot{\tilde{\varepsilon}}_t^{pl}$ and $\dot{\tilde{\varepsilon}}_c^{pl}$ are the equivalent plastic strain rates, θ is the temperature, and $f_i, (i = 1, 2, ...)$ are other predefined field variables.

As shown in Figure 19.6.3–1, when the concrete specimen is unloaded from any point on the strain softening branch of the stress-strain curves, the unloading response is weakened: the elastic stiffness of the material appears to be damaged (or degraded). The degradation of the elastic stiffness is characterized by two damage variables, d_t and d_c, which are assumed to be functions of the plastic strains, temperature, and field variables:

$$d_t = d_t(\tilde{\varepsilon}_t^{pl}, \theta, f_i); \quad 0 \le d_t \le 1,$$
$$d_c = d_c(\tilde{\varepsilon}_c^{pl}, \theta, f_i); \quad 0 \le d_c \le 1.$$

The damage variables can take values from zero, representing the undamaged material, to one, which represents total loss of strength.

If E_0 is the initial (undamaged) elastic stiffness of the material, the stress-strain relations under uniaxial tension and compression loading are, respectively:

$$\sigma_t = (1 - d_t)E_0(\varepsilon_t - \tilde{\varepsilon}_t^{pl}),$$
$$\sigma_c = (1 - d_c)E_0(\varepsilon_c - \tilde{\varepsilon}_c^{pl}).$$

We define the "effective" tensile and compressive cohesion stresses as

$$\bar{\sigma}_t = \frac{\sigma_t}{(1 - d_t)} = E_0(\varepsilon_t - \tilde{\varepsilon}_t^{pl}),$$
$$\bar{\sigma}_c = \frac{\sigma_c}{(1 - d_c)} = E_0(\varepsilon_c - \tilde{\varepsilon}_c^{pl}).$$

The effective cohesion stresses determine the size of the yield (or failure) surface.

Uniaxial cyclic behavior

Under uniaxial cyclic loading conditions the degradation mechanisms are quite complex, involving the opening and closing of previously formed micro-cracks, as well as their interaction. Experimentally, it is observed that there is some recovery of the elastic stiffness as the load changes sign during a uniaxial cyclic test. The stiffness recovery effect, also known as the "unilateral effect," is an important aspect of the concrete behavior under cyclic loading. The effect is usually more pronounced as the load changes from tension to compression, causing tensile cracks to close, which results in the recovery of the compressive stiffness.

The concrete damaged plasticity model assumes that the reduction of the elastic modulus is given in terms of a scalar degradation variable d as

$$E = (1 - d)E_0.$$

where E_0 is the initial (undamaged) modulus of the material.

This expression holds both in the tensile ($\sigma_{11} > 0$) and the compressive ($\sigma_{11} < 0$) sides of the cycle. The stiffness degradation variable, *d*, is a function of the stress state and the uniaxial damage variables, d_t and d_c. For the uniaxial cyclic conditions Abaqus assumes that

$$(1 - d) = (1 - s_t d_c)(1 - s_c d_t).$$

where s_t and s_c are functions of the stress state that are introduced to model stiffness recovery effects associated with stress reversals. They are defined according to

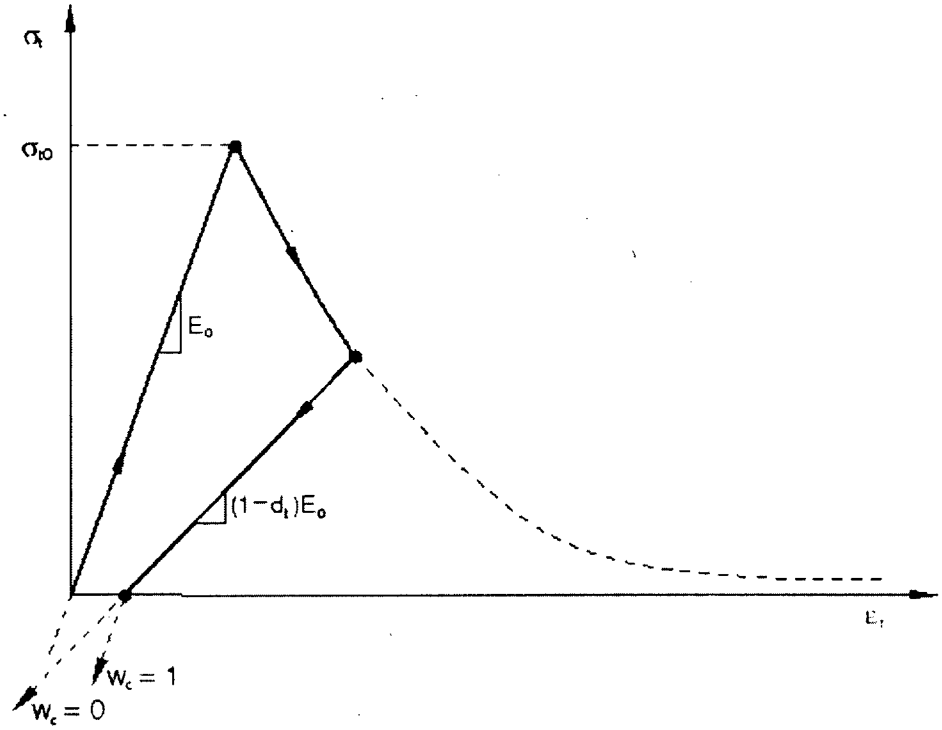
$$s_t = 1 - w_t r^*(\sigma_{11}); \quad 0 \le w_t \le 1,$$
$$s_c = 1 - w_c(1 - r^*(\sigma_{11})); \quad 0 \le w_c \le 1,$$

where

$$r^*(\sigma_{11}) = H(\sigma_{11}) = \begin{cases} 1 & \text{if} \quad \sigma_{11} > 0 \\ 0 & \text{if} \quad \sigma_{11} < 0 \end{cases}$$

The weight factors w_t and w_c, which are assumed to be material properties, control the recovery of the tensile and compressive stiffness upon load reversal. To illustrate this, consider the example in Figure 19.6.3–2, where the load changes from tension to compression.

Figure 19.6.3–2 Illustration of the effect of the compression stiffness recovery parameter w_c.



Assume that there was no previous compressive damage (crushing) in the material; that is, $\tilde{\varepsilon}_c^{pl} = 0$ and $d_c = 0$. Then

$$(1-d) = (1 - s_c d_t) = (1 - (1 - w_c(1-r^*))d_t).$$

- In tension ($\sigma_{11} > 0$), $r^* = 1$; therefore, $d = d_t$ as expected.
- In compression ($\sigma_{11} < 0$), $r^* = 0$, and $d = (1-w_c)d_t$. If $w_c = 1$, then $d = 0$; therefore, the material fully recovers the compressive stiffness (which in this case is the initial undamaged stiffness, $E = E_0$). If, on the other hand, $w_c = 0$, then $d = d_t$ and there is no stiffness recovery. Intermediate values of w_c result in partial recovery of the stiffness.

Multiaxial behavior

The stress-strain relations for the general three-dimensional multiaxial condition are given by the scalar damage elasticity equation:

$$\sigma = (1-d)\mathbf{D}_0^{el} : (\varepsilon - \varepsilon^{pl}).$$

where $\mathbf{D}_0^{el}$ is the initial (undamaged) elasticity matrix.

The previous expression for the scalar stiffness degradation variable, *d*, is generalized to the multiaxial stress case by replacing the unit step function $r^*(\sigma_{11})$ with a multiaxial stress weight factor, $r(\hat{\sigma})$, defined as

$$r(\hat{\sigma}) = \frac{\sum_{i=1}^{3} \langle \hat{\sigma}_i \rangle}{\sum_{i=1}^{3} |\hat{\sigma}_i|}; \quad 0 \le r(\hat{\sigma}) \le 1.$$

where $\hat{\sigma}_i$ $(i = 1, 2, 3)$ are the principal stress components. The Macauley bracket $\langle \cdot \rangle$ is defined by $\langle x \rangle = \frac{1}{2}(|x| + x)$.

See "Damaged plasticity model for concrete and other quasi-brittle materials," Section 4.5.2 of the Abaqus Theory Manual, for further details of the constitutive model.

Reinforcement

In Abaqus reinforcement in concrete structures is typically provided by means of rebars, which are one-dimensional rods that can be defined singly or embedded in oriented surfaces. Rebars are typically used with metal plasticity models to describe the behavior of the rebar material and are superposed on a mesh of standard element types used to model the concrete.

With this modeling approach, the concrete behavior is considered independently of the rebar. Effects associated with the rebar/concrete interface, such as bond slip and dowel action, are modeled approximately by introducing some "tension stiffening" into the concrete modeling to simulate load transfer across cracks through the rebar. Details regarding tension stiffening are provided below.

Defining the rebar can be tedious in complex problems, but it is important that this be done accurately since it may cause an analysis to fail due to lack of reinforcement in key regions of a model. See "Defining rebar as an element property," Section 2.2.4, for more information regarding rebars.

Defining tension stiffening

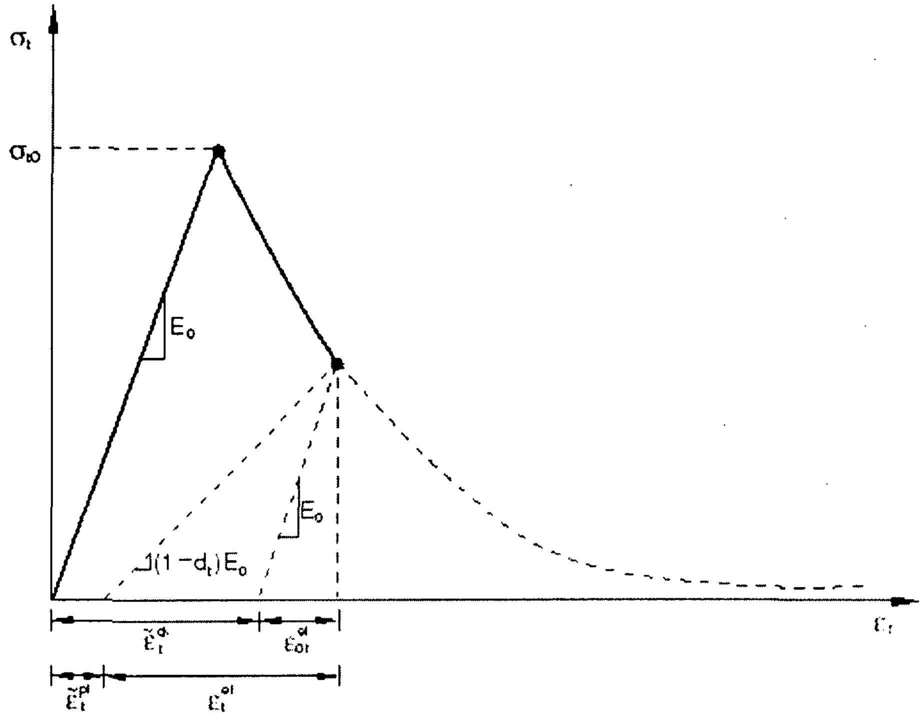
The postfailure behavior for direct straining is modeled with tension stiffening, which allows you to define the strain-softening behavior for cracked concrete. This behavior also allows for the effects of the reinforcement interaction with concrete to be simulated in a simple manner. Tension stiffening is required in the concrete damaged plasticity model. You can specify tension stiffening by means of a postfailure stress-strain relation or by applying a fracture energy cracking criterion.

Postfailure stress-strain relation

In reinforced concrete the specification of postfailure behavior generally means giving the postfailure stress as a function of cracking strain, $\tilde{\varepsilon}_t^{ck}$. The cracking strain is defined as the total strain minus the elastic strain corresponding to the undamaged material;

that is, $\tilde{\varepsilon}_t^{ck} = \varepsilon_t - \varepsilon_{0t}^{el}$, where $\varepsilon_{0t}^{el} = \sigma_t / E_0$, as illustrated in Figure 19.6.3–3. To avoid potential numerical problems, Abaqus enforces a lower limit on the postfailure stress equal to one hundred of the initial failure stress: $\sigma_t \geq \sigma_{t0}/100$.

Figure 19.6.3–3 Illustration of the definition of the cracking strain $\tilde{\varepsilon}_t^{ck}$ used for the definition of tension stiffening data.



Tension stiffening data are given in terms of the cracking strain, $\tilde{\varepsilon}_t^{ck}$. When unloading data are available, the data are provided to Abaqus in terms of tensile damage curves, $d_t - \tilde{\varepsilon}_t^{ck}$, as discussed below. Abaqus automatically converts the cracking strain values to plastic strain values using the relationship

$$\tilde{\varepsilon}_t^{pl} = \tilde{\varepsilon}_t^{ck} - \frac{d_t}{(1-d_t)} \frac{\sigma_t}{E_0}.$$

Abaqus will issue an error message if the calculated plastic strain values are negative and/or decreasing with increasing cracking strain, which typically indicates that the tensile damage curves are incorrect. In the absence of tensile damage $\tilde{\varepsilon}_t^{pl} = \tilde{\varepsilon}_t^{ck}$.

In cases with little or no reinforcement, the specification of a postfailure stress-strain relation introduces mesh sensitivity in the results, in the sense that the finite element predictions do not converge to a unique solution as the mesh is refined because mesh refinement leads to narrower crack bands. This problem typically occurs if cracking failure occurs only at localized regions in the structure and mesh refinement does not result in the formation of additional cracks. If cracking failure is distributed evenly (either due to the effect of rebar or due to the presence of stabilizing elastic material, as in the case of plate bending), mesh sensitivity is less of a concern.

In practical calculations for reinforced concrete, the mesh is usually such that each element contains rebars. The interaction between the rebars and the concrete tends to reduce the mesh sensitivity, provided that a reasonable amount of tension stiffening is introduced in the concrete model to simulate this interaction. This requires an estimate of the tension stiffening effect, which depends on such factors as the density of reinforcement, the quality of the bond between the rebar and the concrete, the relative size of the concrete aggregate compared to the rebar diameter, and the mesh. A reasonable starting point for relatively heavily reinforced concrete modeled with a fairly detailed mesh is to assume that the strain softening after failure reduces the stress linearly to zero at a total strain of about 10 times the strain at failure. The strain at failure in standard concretes is typically 10^{-4}, which suggests that tension stiffening that reduces the stress to zero at a total strain of about 10^{-3} is reasonable. This parameter should be calibrated to a particular case.

The choice of tension stiffening parameters is important since, generally, more tension stiffening makes it easier to obtain numerical solutions. Too little tension stiffening will cause the local cracking failure in the concrete to introduce temporarily unstable behavior in the overall response of the model. Few practical designs exhibit such behavior, so that the presence of this type of response in the analysis model usually indicates that the tension stiffening is unreasonably low.

Input File Usage: *CONCRETE TENSION STIFFENING, TYPE=STRAIN (default)

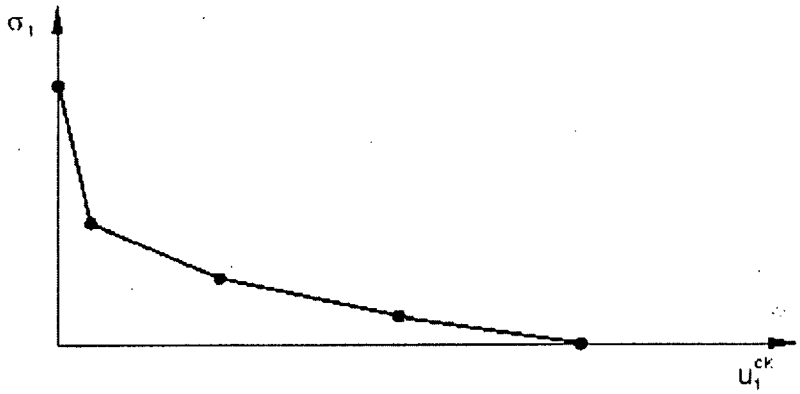
Abaqus/CAE Usage: Property module: material editor: **Mechanical⟶Plasticity⟶ Concrete Damaged Plasticity: Tensile Behavior: Type: Strain**

Fracture energy cracking criterion

When there is no reinforcement in significant regions of the model, the tension stiffening approach described above will introduce unreasonable mesh sensitivity into the results. However, it is generally accepted that Hillerborg's (1976) fracture energy proposal is adequate to allay the concern for many practical purposes. Hillerborg defines the energy required to open a unit area of crack, G_f, as a material parameter, using brittle fracture concepts. With this approach the concrete's brittle behavior is characterized by a stress-displacement response rather than a stress-strain response. Under tension a concrete specimen will crack across some section. After it has been pulled apart sufficiently for most of the stress to be removed (so that the undamaged elastic strain is small), its length will be determined primarily by the opening at the crack. The opening does not depend on the specimen's length.

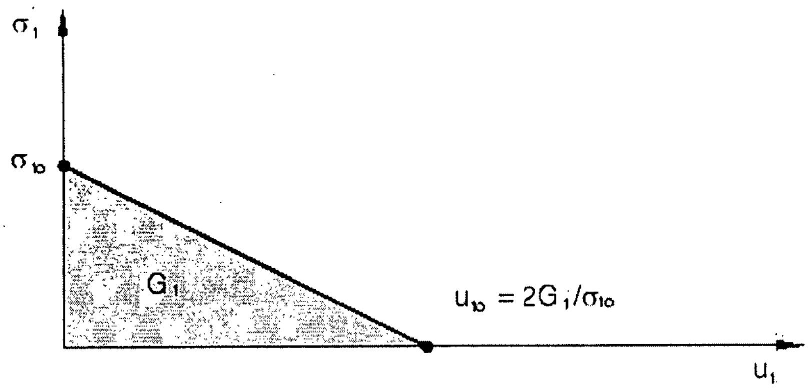
This fracture energy cracking model can be invoked by specifying the postfailure stress as a tabular function of cracking displacement, as shown in Figure 19.6.3–4.

Figure 19.6.3–4 Postfailure stress-displacement curve.



Alternatively, the fracture energy, G_f, can be specified directly as a material property; in this case, define the failure stress, σ_{t0}, as a tabular function of the associated fracture energy. This model assumes a linear loss of strength after cracking, as shown in Figure 19.6.3–5.

Figure 19.6.3–5 Postfailure stress-fracture energy curve.



The cracking displacement at which complete loss of strength takes place is, therefore, $u_{t0} = 2G_f/\sigma_{t0}$. Typical values of G_f range from 40 N/m (0.22 lb/in) for a typical construction concrete (with a compressive strength of approximately 20 MPa, 2850 lb/in^2) to 120 N/m (0.67 lb/in) for a high-strength concrete (with a compressive strength of approximately 40 MPa, 5700 lb/in^2).

If tensile damage, d_t, is specified, Abaqus automatically converts the cracking displacement values to "plastic" displacement values using the relationship

$$u_t^{pl} = u_t^{ck} - \frac{d_t}{(1-d_t)} \frac{\sigma_t l_0}{E_0}.$$

where the specimen length, l_0, is assumed to be one unit length, $l_0 = 1$.

Implementation

The implementation of this stress-displacement concept in a finite element model requires the definition of a characteristic length associated with an integration point. The characteristic crack length is based on the element geometry and formulation: it is a typical length of a line across an element for a first-order element; it is half of the same typical length for a second-order element. For beams and trusses it is a characteristic length along the element axis. For membranes and shells it is a characteristic length in the reference surface. For axisymmetric elements it is a characteristic length in the *r–z* plane only. For cohesive elements it is equal to the constitutive thickness. This definition of the characteristic crack length is used because the direction in which cracking occurs is not known in advance. Therefore, elements with large aspect ratios will have rather different behavior depending on the direction in which they crack: some mesh sensitivity remains because of this effect, and elements that have aspect ratios close to one are recommended.

Input File Usage: Use the following option to specify the postfailure stress as a tabular function of displacement:
*CONCRETE TENSION STIFFENING, TYPE=DISPLACEMENT

Use the following option to specify the postfailure stress as a tabular function of the fracture energy:

*CONCRETE TENSION STIFFENING, TYPE=GFI

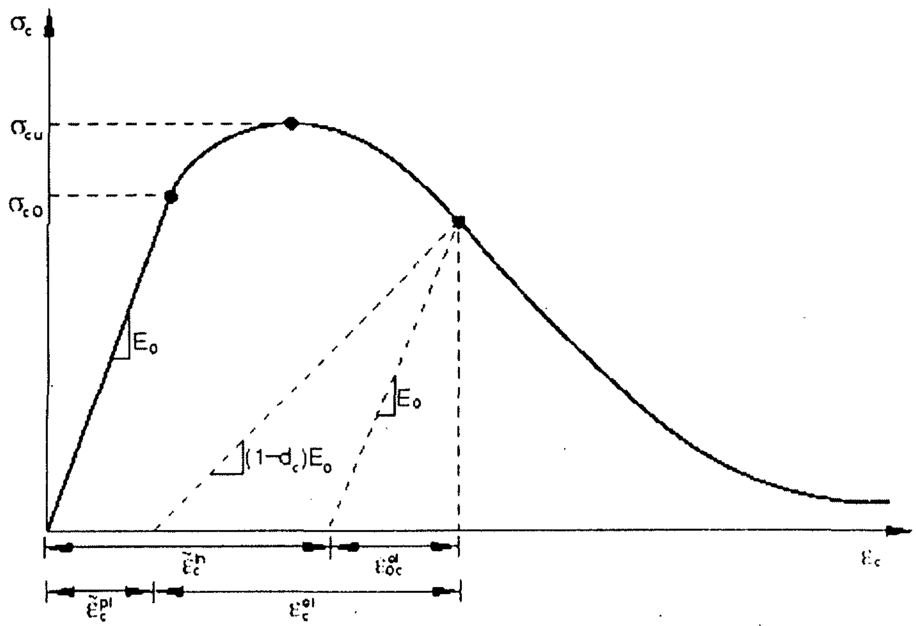
Abaqus/CAE Usage: Property module: material editor: **Mechanical⟶Plasticity⟶ Concrete Damaged Plasticity**: **Tensile Behavior**: **Type: Displacement** or **GFI**

Defining compressive behavior

You can define the stress-strain behavior of plain concrete in uniaxial compression outside the elastic range. Compressive stress data are provided as a tabular function of inelastic (or crushing) strain, $\tilde{\varepsilon}_c^{in}$, and, if desired, strain rate, temperature, and field variables. Positive (absolute) values should be given for the compressive stress and strain. The stress-strain curve can be defined beyond the ultimate stress, into the strain-softening regime.

Hardening data are given in terms of an inelastic strain, $\tilde{\varepsilon}_c^{in}$, instead of plastic strain, $\tilde{\varepsilon}_c^{pl}$. The compressive inelastic strain is defined as the total strain minus the elastic strain corresponding to the undamaged material, $\tilde{\varepsilon}_c^{in} = \varepsilon_c - \varepsilon_{0c}^{el}$, where $\varepsilon_{0c}^{el} = \sigma_c / E_0$, as illustrated in Figure 19.6.3–6.

Figure 19.6.3–6 Definition of the compressive inelastic (or crushing) strain $\tilde{\varepsilon}_c^{in}$ used for the definition of compression hardening data.



Unloading data are provided to Abaqus in terms of compressive damage curves, $d_c - \tilde{\varepsilon}_c^{in}$, as discussed below. Abaqus automatically converts the inelastic strain values to plastic strain values using the relationship

$$\tilde{\varepsilon}_c^{pl} = \tilde{\varepsilon}_c^{in} - \frac{d_c}{(1-d_c)} \frac{\sigma_c}{E_0}.$$

Abaqus will issue an error message if the calculated plastic strain values are negative and/or decreasing with increasing inelastic strain, which typically indicates that the compressive damage curves are incorrect. In the absence of compressive damage $\tilde{\varepsilon}_c^{pl} = \tilde{\varepsilon}_c^{in}$.

Input File Usage: *CONCRETE COMPRESSION HARDENING

Abaqus/CAE Usage: Property module: material editor: **Mechanical⟶Plasticity⟶**

Concrete Damaged Plasticity: Compressive Behavior

Defining damage and stiffness recovery

Damage, d_t and/or d_c, can be specified in tabular form. (If damage is not specified, the model behaves as a plasticity model; consequently, $\tilde{\varepsilon}_t^{pl} = \tilde{\varepsilon}_t^{ck}$ and $\tilde{\varepsilon}_c^{pl} = \tilde{\varepsilon}_c^{in}$.)

In Abaqus the damage variables are treated as non-decreasing material point quantities. At any increment during the analysis, the new value of each damage variable is obtained as the maximum between the value at the end of the previous increment and the value corresponding to the current state (interpolated from the user-specified tabular data); that is,

$$d_t|_{t+\Delta t} = \max\left\{d_t|_t, d_t(\tilde{\varepsilon}_t^{pl}|_{t+\Delta t}, \theta|_{t+\Delta t}, f_i|_{t+\Delta t})\right\},$$

$$d_c|_{t+\Delta t} = \max\left\{d_c|_t, d_c(\tilde{\varepsilon}_c^{pl}|_{t+\Delta t}, \theta|_{t+\Delta t}, f_i|_{t+\Delta t})\right\}.$$

The choice of the damage properties is important since, generally, excessive damage may have a critical effect on the rate of convergence. It is recommended to avoid using values of the damage variables above 0.99, which corresponds to a 99% reduction of the stiffness.

Tensile damage

You can define the uniaxial tension damage variable, d_t, as a tabular function of either cracking strain or cracking displacement.

Input File Usage: Use the following option to specify tensile damage as a function of cracking strain:

*CONCRETE TENSION DAMAGE, TYPE=STRAIN (default)

Use the following option to specify tensile damage as a function of cracking displacement:

*CONCRETE TENSION DAMAGE, TYPE=DISPLACEMENT

Abaqus/CAE Usage: Property module: material editor: **Mechanical→Plasticity→Concrete Damaged Plasticity**: **Tensile Behavior**: **Suboptions→Tension Damage**: **Type: Strain** or **Displacement**

Compressive damage

You can define the uniaxial compression damage variable, d_c, as a tabular function of inelastic (crushing) strain.

Input File Usage: *CONCRETE COMPRESSION DAMAGE

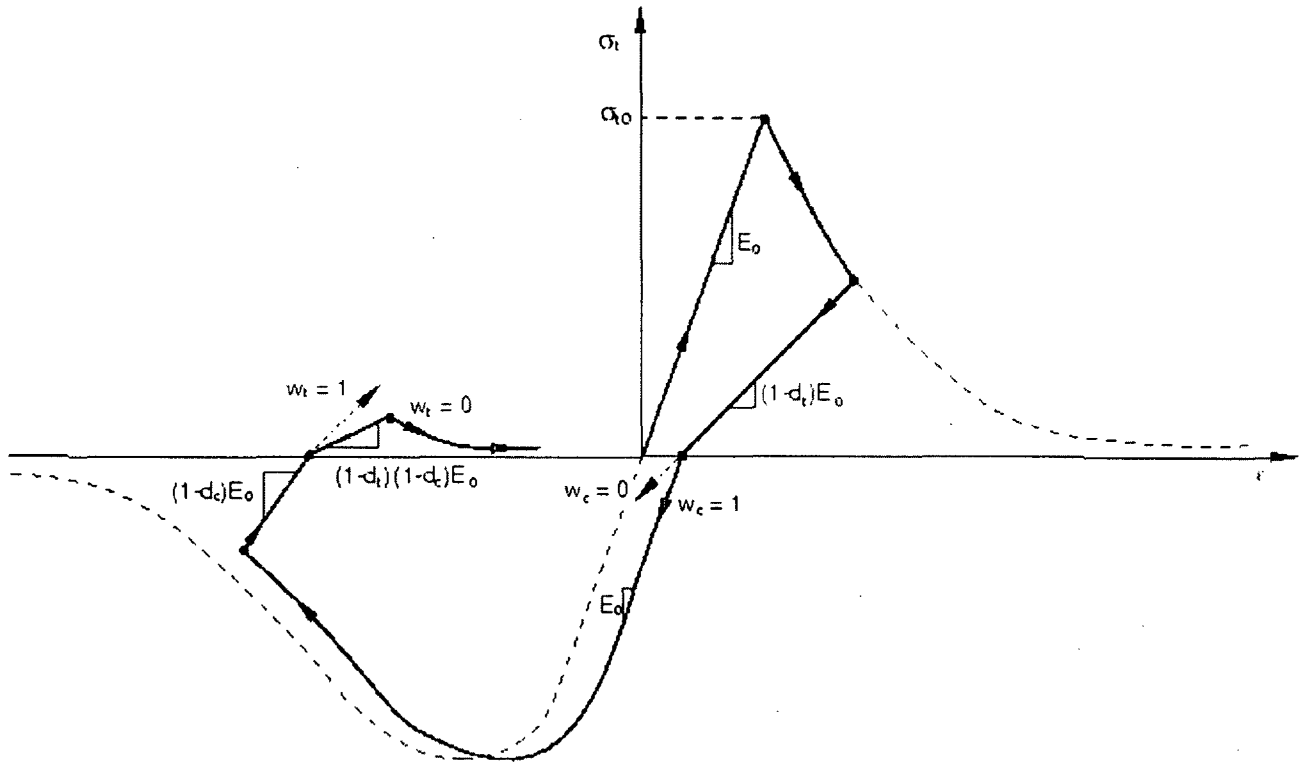
Abaqus/CAE Usage: Property module: material editor: **Mechanical⟶Plasticity⟶Concrete Damaged Plasticity**: **Compressive Behavior**: **Suboptions⟶Compression Damage**

Stiffness recovery

As discussed above, stiffness recovery is an important aspect of the mechanical response of concrete under cyclic loading. Abaqus allows direct user specification of the stiffness recovery factors w_t and w_c.

The experimental observation in most quasi-brittle materials, including concrete, is that the compressive stiffness is recovered upon crack closure as the load changes from tension to compression. On the other hand, the tensile stiffness is not recovered as the load changes from compression to tension once crushing micro-cracks have developed. This behavior, which corresponds to $w_t = 0$ and $w_c = 1$, is the default used by Abaqus. Figure 19.6.3–7 illustrates a uniaxial load cycle assuming the default behavior.

Figure 19.6.3–7 Uniaxial load cycle (tension-compression-tension) assuming default values for the stiffness recovery factors: $w_t = 0$ and $w_c = 1$.



Input File Usage: Use the following option to specify the compression stiffness recovery factor, w_c:

<u>*CONCRETE TENSION DAMAGE</u>, COMPRESSION RECOVERY=w_c

Use the following option to specify the tension stiffness recovery factor, w_t:

<u>*CONCRETE COMPRESSION DAMAGE</u>, TENSION RECOVERY=w_t

Abaqus/CAE Usage: Property module: material editor: **Mechanical⟶Plasticity⟶ Concrete Damaged Plasticity**: **Tensile Behavior**: **Suboptions⟶Tension Damage**: **Compression recovery:** w_c **Compressive Behavior**: **Suboptions⟶Compression Damage**: **Tension recovery:** w_t

Rate dependence

The rate-sensitive behavior of quasi-brittle materials is mainly connected to the retardation effects that high strain rates have on the growth of micro-cracks. The effect is usually more pronounced under tensile loading. As the strain rate increases, the stress-strain curves exhibit decreasing nonlinearity as well as an increase in the peak strength. You can specify tension stiffening as a tabular function of cracking strain (or displacement) rate, and you can specify compression hardening data as a tabular function of inelastic strain rate.

Input File Usage: Use the following options:
*CONCRETE TENSION STIFFENING
*CONCRETE COMPRESSION HARDENING

Abaqus/CAE Usage: Property module: material editor: **Mechanical⟶Plasticity⟶ Concrete Damaged Plasticity**: **Tensile Behavior**: **Use strain-rate-dependent data Compressive Behavior**: **Use strain-rate-dependent data**

Concrete plasticity

You can define flow potential, yield surface, and in Abaqus/Standard viscosity parameters for the concrete damaged plasticity material model.

Input File Usage: *CONCRETE DAMAGED PLASTICITY

Abaqus/CAE Usage: Property module: material editor: **Mechanical⟶Plasticity⟶ Concrete Damaged Plasticity**: **Plasticity**

Effective stress invariants

The effective stress is defined as

$$\bar{\sigma} = \mathbf{D}_0^{el} : (\varepsilon - \varepsilon^{pl}).$$

The plastic flow potential function and the yield surface make use of two stress invariants of the effective stress tensor, namely the hydrostatic pressure stress,

$$\bar{p} = -\frac{1}{3}\operatorname{trace}(\bar{\sigma}),$$

and the Mises equivalent effective stress,

$$\bar{q} = \sqrt{\frac{3}{2}(\bar{\mathbf{S}} : \bar{\mathbf{S}})},$$

where $\bar{\mathbf{S}}$is the effective stress deviator, defined as

$$\bar{\mathbf{S}} = \bar{\sigma} + \bar{p}\mathbf{I}.$$

Plastic flow

The concrete damaged plasticity model assumes nonassociated potential plastic flow. The flow potential *G* used for this model is the Drucker-Prager hyperbolic function:

$$G = \sqrt{(\epsilon\sigma_{t0}\tan\psi)^2 + \bar{q}^2} - \bar{p}\tan\psi.$$

where

$\psi(\theta, f_i)$

is the dilation angle measured in the *p*–*q* plane at high confining pressure;

$\sigma_{t0}(\theta, f_i) = \sigma_t|_{\tilde{\varepsilon}_t^{pl}=0,\dot{\tilde{\varepsilon}}_t^{pl}=0}$

is the uniaxial tensile stress at failure, taken from the user-specified tension stiffening data; and

$\epsilon(\theta, f_i)$

is a parameter, referred to as the eccentricity, that defines the rate at which the function approaches the asymptote (the flow potential tends to a straight line as the eccentricity tends to zero).

This flow potential, which is continuous and smooth, ensures that the flow direction is always uniquely defined. The function approaches the linear Drucker-Prager flow potential asymptotically at high confining pressure stress and intersects the hydrostatic pressure axis at 90?. See "Models for granular or polymer behavior," Section 4.4.2 of the Abaqus Theory Manual, for further discussion of this potential.

The default flow potential eccentricity is $\epsilon = 0.1$, which implies that the material has almost the same dilation angle over a wide range of confining pressure stress values. Increasing the value of ϵ provides more curvature to the flow potential, implying that the dilation angle increases more rapidly as the confining pressure decreases. Values of ϵ that are significantly less than the default value may lead to convergence problems if the material is subjected to low confining pressures because of the very tight curvature of the flow potential locally where it intersects the *p*-axis.

Yield function

The model makes use of the yield function of Lubliner et. al. (1989), with the modifications proposed by Lee and Fenves (1998) to account for different evolution of strength under tension and compression. The evolution of the yield surface is controlled by the hardening variables, $\tilde{\varepsilon}_t^{pl}$ and $\tilde{\varepsilon}_c^{pl}$. In terms of effective stresses, the yield function takes the form

$$F = \frac{1}{1-\alpha}\left(\bar{q} - 3\alpha\bar{p} + \beta(\tilde{\varepsilon}^{pl})\langle\hat{\bar{\sigma}}_{\max}\rangle - \gamma\langle-\hat{\bar{\sigma}}_{\max}\rangle\right) - \bar{\sigma}_c(\tilde{\varepsilon}_c^{pl}) = 0,$$

with

$$\alpha = \frac{(\sigma_{b0}/\sigma_{c0}) - 1}{2(\sigma_{b0}/\sigma_{c0}) - 1};\ 0 \le \alpha \le 0.5,$$

$$\beta = \frac{\bar{\sigma}_c(\tilde{\varepsilon}_c^{pl})}{\bar{\sigma}_t(\tilde{\varepsilon}_t^{pl})}(1-\alpha) - (1+\alpha),$$

$$\gamma = \frac{3(1-K_c)}{2K_c - 1}.$$

Here,

$\hat{\bar{\sigma}}_{\max}$

is the maximum principal effective stress;

σ_{b0}/σ_{c0}

is the ratio of initial equibiaxial compressive yield stress to initial uniaxial compressive yield stress (the default value is 1.16);

K_c

is the ratio of the second stress invariant on the tensile meridian, $q_{(TM)}$, to that on the compressive meridian, $q_{(CM)}$, at initial yield for any given value of the pressure invariant *p* such that the maximum principal stress is negative, $\hat{\sigma}_{\max} < 0$(see Figure 19.6.3–8); it must satisfy the condition $0.5 < K_c \le 1.0$(the default value is $2/3$);

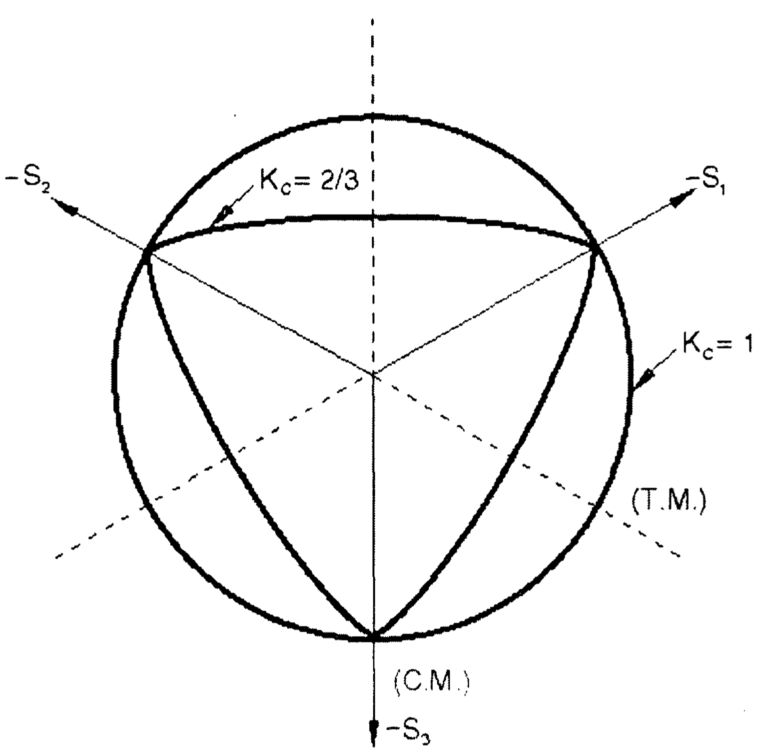
$\bar{\sigma}_t(\tilde{\varepsilon}_t^{pl})$

is the effective tensile cohesion stress; and

$\bar{\sigma}_c(\tilde{\varepsilon}_c^{pl})$

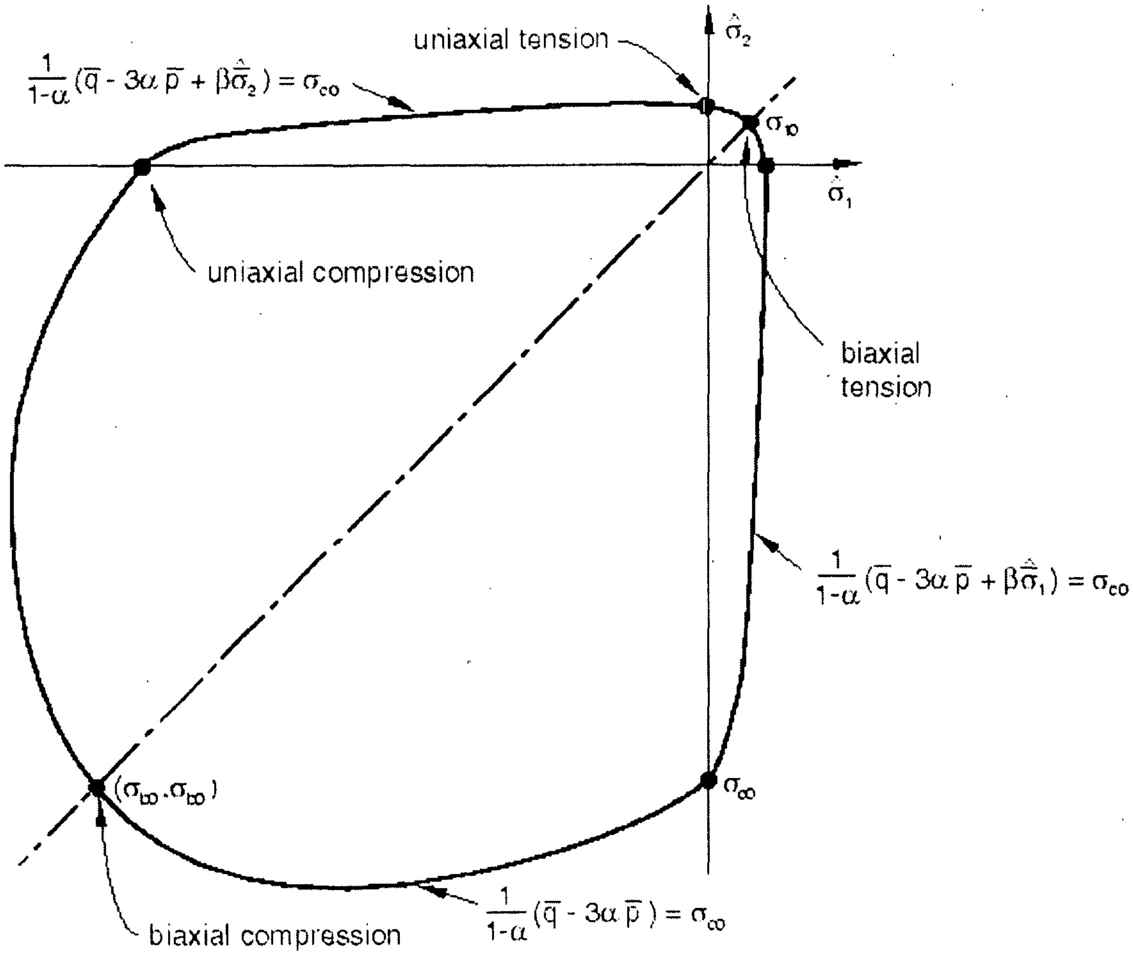
is the effective compressive cohesion stress.

Figure 19.6.3–8 Yield surfaces in the deviatoric plane, corresponding to different values of K_c.



Typical yield surfaces are shown in Figure 19.6.3–8 on the deviatoric plane and in Figure 19.6.3–9 for plane stress conditions.

Figure 19.6.3–9 Yield surface in plane stress.



Nonassociated flow

Because plastic flow is nonassociated, the use of concrete damaged plasticity results in a nonsymmetric material stiffness matrix. Therefore, to obtain an acceptable rate of convergence in Abaqus/Standard, the unsymmetric matrix storage and solution scheme should be used. Abaqus/Standard will automatically activate the unsymmetric solution scheme if concrete damaged plasticity is used in the analysis. If desired, you can turn off the unsymmetric solution scheme for a particular step (see "Procedures: overview," Section 6.1.1).

Viscoplastic regularization

Material models exhibiting softening behavior and stiffness degradation often lead to severe convergence difficulties in implicit analysis programs, such as Abaqus/Standard. A common technique to overcome some of these convergence difficulties is the use of a viscoplastic regularization of the constitutive equations, which causes the consistent tangent stiffness of the softening material to become positive for sufficiently small time increments.

The concrete damaged plasticity model can be regularized in Abaqus/Standard using viscoplasticity by permitting stresses to be outside of the yield surface. We use a generalization of the Duvaut-Lions regularization, according to which the viscoplastic strain rate tensor, $\dot{\varepsilon}_v^{pl}$, is defined as

$$\dot{\varepsilon}_v^{pl} = \frac{1}{\mu}(\varepsilon^{pl} - \varepsilon_v^{pl}).$$

Here μ is the viscosity parameter representing the relaxation time of the viscoplastic system, and ε^{pl} is the plastic strain evaluated in the inviscid backbone model.

Similarly, a viscous stiffness degradation variable, d_v, for the viscoplastic system is defined as

$$\dot{d}_v = \frac{1}{\mu}(d - d_v),$$

where *d* is the degradation variable evaluated in the inviscid backbone model. The stress-strain relation of the viscoplastic model is given as

$$\boldsymbol{\sigma} = (1 - d_v)\mathbf{D}_0^{el} : (\boldsymbol{\varepsilon} - \boldsymbol{\varepsilon}_v^{pl}).$$

Using the viscoplastic regularization with a small value for the viscosity parameter (small compared to the characteristic time increment) usually helps improve the rate of convergence of the model in the softening regime, without compromising results. The basic idea is that the solution of the viscoplastic system relaxes to that of the inviscid case as $t/\mu \rightarrow \infty$, where *t* represents time. You can specify the value of the viscosity parameter as part of the concrete damaged plasticity material behavior definition. If the viscosity parameter is different from zero, output results of the plastic strain and stiffness degradation refer to the viscoplastic values, ε_v^{pl} and d_v. In Abaqus/Standard the default value of the viscosity parameter is zero, so that no viscoplastic regularization is performed.

Material damping

The concrete damaged plasticity model can be used in combination with material damping (see "Material damping," Section 22.1.1). If stiffness proportional damping is specified, Abaqus calculates the damping stress based on the undamaged elastic stiffness. This may introduce large artificial damping forces on elements undergoing severe damage at high strain rates.

Visualization of "crack directions"

Unlike concrete models based on the smeared crack approach, the concrete damaged plasticity model does not have the notion of cracks developing at the material integration point. However, it is possible to introduce the concept of an effective crack direction with the purpose of obtaining a graphical visualization of the cracking patterns in the concrete structure. Different criteria can be adopted within the framework of scalar-damage plasticity for the definition of the direction of cracking. Following Lubliner et. al. (1989), we can assume that cracking initiates at points where the tensile equivalent plastic strain is greater than zero, $\tilde{\varepsilon}_t^{pl} > 0$, and the maximum principal plastic strain is positive. The direction of the vector normal to the crack plane is assumed to be parallel to the direction of the maximum principal plastic strain. This direction can be viewed in the Visualization module of Abaqus/CAE.

Abaqus/CAE Usage: Visualization module: **Result⟶Field Output**: **PE, Max. Principal Plot⟶Symbols**

Elements

Abaqus offers a variety of elements for use with the concrete damaged plasticity model: truss, shell, plane stress, plane strain, generalized plane strain, axisymmetric, and three-dimensional elements. Most beam elements can be used; however, beam elements in space that include shear stress caused by torsion and do not include hoop stress (such as B31, B31H, B32, B32H, B33, and B33H) cannot be used. Thin-walled, open-section beam elements and PIPE elements can be used with the concrete damaged plasticity model.

For general shell analysis more than the default number of five integration points through the thickness of the shell should be used; nine thickness integration points are commonly used to model progressive failure of the concrete through the thickness with acceptable accuracy.

Output

In addition to the standard output identifiers available in Abaqus ("Abaqus/Standard output variable identifiers," Section 4.2.1, and "Abaqus/Explicit output variable identifiers," Section 4.2.2), the following variables relate specifically to material points in the concrete damaged plasticity model:

DAMAGEC Compressive damage variable, d_c.

DAMAGET Tensile damage variable, d_t.

PEEQ Compressive equivalent plastic strain, $\tilde{\varepsilon}_c^{pl}$.

PEEQT Tensile equivalent plastic strain, $\tilde{\varepsilon}_t^{pl}$.

SDEG Stiffness degradation variable, *d*.

DMENER Energy dissipated per unit volume by damage.

ELDMD Total energy dissipated in the element by damage.

ALLDMD Energy dissipated in the whole (or partial) model by damage. The contribution from ALLDMD is included in the total strain energy ALLIE.

EDMDDEN Energy dissipated per unit volume in the element by damage.

SENER The recoverable part of the energy per unit volume.

ELSE The recoverable part of the energy in the element.

ALLSE The recoverable part of the energy in the whole (partial) model.

ESEDEN The recoverable part of the energy per unit volume in the element.

Additional references

- Hillerborg, A., M. Modeer, and P. E. Petersson, “Analysis of Crack Formation and Crack Growth in Concrete by Means of Fracture Mechanics and Finite Elements,” Cement and Concrete Research, vol. 6, pp. 773–782, 1976.
- Lee, J., and G. L. Fenves, “Plastic-Damage Model for Cyclic Loading of Concrete Structures,” Journal of Engineering Mechanics, vol. 124, no.8, pp. 892–900, 1998.
- Lubliner, J., J. Oliver, S. Oller, and E. O□, “A Plastic-Damage Model for Concrete,” International Journal of Solids and Structures, vol. 25, pp. 299–329, 1989.